"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"

Dissertation submitted to

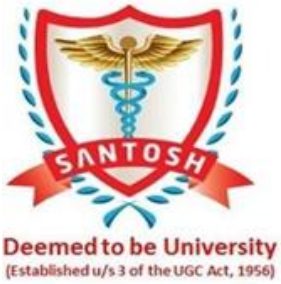
SANTOSH DEEMED TO BE UNIVERSITY, GHAZIABAD, NCR, DELHI

In partial fulfillment of the requirement for the degree of

MASTER OF DENTAL SURGERY

In the Specialty of

PROSTHODONTICS AND CROWN & BRIDGE



By

Dr. AMBIKA VERMA

Under the guidance of

Dr. RAJIV KUMAR GUPTA

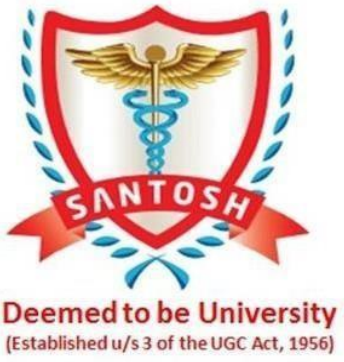
Professor & Head

DEPARTMENT OF PROSTHODONTICS AND CROWN & BRIDGE

SANTOSH DENTAL COLLEGE & HOSPITALS

GHAZIABAD- 201009, NCR, DELHI, INDIA

2022-2025



COPYRIGHT

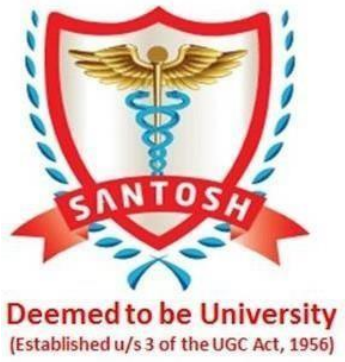
DECLARATION BY THE CANDIDATE

I hereby, declare that the Santosh Deemed to be University, Ghaziabad, NCR, Delhi shall have the right to preserve, use and disseminate this dissertation in print or electronic format for academic or research purpose.

Date: **DR. AMBIKA VERMA**

Place: GHAZIABAD

© Santosh Deemed to be University, Ghaziabad, Uttar Pradesh



DECLARATION BY THE CANDIDATE

I hereby, declare that this dissertation entitled **"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"** is a Bonafide genuine research carried out by me under the esteemed able guidance of my guide **Prof. Dr. Rajiv Kumar Gupta (MDS) Professor & Head**, and Co-guide **Dr. Bharti Dua (MDS) Reader**, Department of Prosthodontics and Crown & Bridge, Santosh Dental College & Hospitals, Ghaziabad, NCR, Delhi.

Date: **Dr. AMBIKA VERMA**

Place: Ghaziabad

Deemed to be University
(Established u/s 3 of the UGC Act, 1956)

ENDORSEMENT BY THE PRINCIPAL/ DEAN OF THE INSTITUTION

This is to certify that the dissertation entitled **"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"**, is a Bonafide research work done by **Dr. Ambika Verma** herself under the esteemed guidance of (Prof.) **Dr. Akshay Bhargava**, Dean, Department of Prosthodontics and Crown & Bridge, Santosh Dental College and Hospitals Ghaziabad, NCR, Delhi.

Date:

Place: Ghaziabad

(Prof.) Dr. AKSHAY BHARGAVA
MDS (Prosthodontics)
AFAAID, DICOI, DWCOI
Professor & Dean
Santosh Dental College & Hospital
GHAZIABAD, NCR, DELHI

Deemed to be University
(Established u/s 3 of the UGC Act, 1956)

CERTIFICATE BY THE GUIDE AND CO-GUIDE

This is to certify that the dissertation **"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"** is a Bonafide genuine research work carried out by **Dr. Ambika Verma**, in partial fulfillment of the requirement for the degree of Master of Dental Surgery in the specialty of Prosthodontics and Crown & Bridge.

GUIDE

(Prof.) Dr. RAJIV KUMAR GUPTA
HOD
MDS (Prosthodontics)

CO-GUIDE

(Prof.) Dr. BHARTI DUA
READER
MDS (Prosthodontics)

Date:

Place:

Department of Prosthodontics and Crown & Bridge
Santosh Dental College & Hospital

Ghaziabad, NCR, Delhi

Deemed to be University
(Established u/s 3 of the UGC Act, 1956)

CERTIFICATE BY HEAD OF THE DEPARTMENT

This is to certify that the Dissertation entitled **"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"** being submitted by **Dr. Ambika Verma** in partial fulfillment of regulation for the award of the degree of Master of Dental Surgery in the specialty of Prosthodontics and Crown & Bridge is based on the bonafide work carried out by her under my direct guidance and supervision and completed to my full satisfaction.

Date

Place: Ghaziabad **HEAD OF DEPARTMENT**

Dr. RAJIV KUMAR GUPTA
M.D.S. (Prostho),
Professor & HOD
Department of Prosthodontics and Crown & Bridge
Santosh Dental College & Hospitals
Ghaziabad, NCR, Delhi

Deemed to be University
(Established u/s 3 of the UGC Act, 1956)

ANTIPLAGIARISM DECLARATION/UNDERTAKING

I, **Dr. Ambika Verma,** hereby declare that the present thesis entitled, **"A FINITE ELEMENT ANALYSIS STUDY TO EVALUATE THE EFFECT OF IMPLANT TILT AND TRANSMUCOSAL ABUTMENT ANGULATION AS INFLUENCING THE STRESS PATTERN"** submitted by me to Santosh Deemed to be University is a bonafide/ original research work. I also hereby declare that the similarity index is less than ten percent and the present research work has not been submitted for the award of any degree/diploma of the same institution where the work was carried out or to any other Institution.

(Dr. AMBIKA VERMA)

I have verified the contents of the thesis, and the similarity index is less than ten percent.

Dr. Jyoti Batra
(Dean Research)

Date:

Place: Santosh Deemed to be University

ACKNOWLEDGEMENTS

Acknowledgment is a humble expression of appreciation—an offering of heartfelt thanks to all those whose presence, guidance, and unwavering support breathe life into any academic endeavor. This dissertation is not merely the result of personal effort, but the culmination of the kindness, wisdom, and encouragement extended to me by many individuals. With deep reverence and gratitude, I dedicate this section to those who walked beside me during this journey, in both seen and unseen ways.

First and foremost, I bow my head in gratitude to the Almighty, the most merciful and compassionate, who has been my constant source of strength, courage, and clarity. In moments of uncertainty and fatigue, it was His divine grace that renewed my spirit and allowed me to persevere.

It is my absolute honor and privilege to express my deepest gratitude to my esteemed guide and mentor, ***(Prof.) Dr. Rajiv Gupta, Head of the Department of Prosthodontics and Crown & Bridge, Santosh Dental College and Hospital.*** *Sir, your profound knowledge, relentless pursuit of excellence, and meticulous attention to every detail have been nothing short of inspiring. Your patience, integrity, and encouragement—even amidst your demanding schedule—shaped not just this work, but also my growth as a scholar and clinician. I am forever indebted to your guidance and mentorship, which have left an indelible mark on my academic journey.*

My heartfelt thanks ***to (Prof.) Dr. Akshay Bhargava, Dean, Santosh Dental College and Hospital,*** *for his gracious support, scholarly wisdom, and for enabling the academic environment and resources that made this research possible. His passion for innovation and learning has been a source of great inspiration.*

I also extend my sincere appreciation to my co-guide, ***Dr. Bharti Dua,*** *Department of Prosthodontics, for her thoughtful suggestions and kind guidance throughout the course of this study. I am equally grateful to* ***(Prof.) Dr. Manoj Goyat****,* ***Dr. Kavita****, and* ***Dr. Cherry*** *for their generous support, insightful feedback, and constant encouragement, which contributed immensely to the success of this dissertation.*

No words can truly capture the depth of my gratitude to my parents, ***Mrs. Omwati Verma and Mr. Vidhya Ram.*** *You are the foundation of all that I am and all that I strive to be. Your sacrifices, unconditional love, and unwavering belief in me have been the guiding light through every storm. You held me up when I doubted myself, and celebrated every small victory as if it were your own. This achievement belongs as much to you as it does to me.*

To ***Ishan Gupta,*** *my partner in every sense of the word—thank you for being my strength, my calm, and my unwavering support system. In moments of exhaustion and self-doubt, your patience, love, and steady reassurance reminded me of my potential and gave me the courage to keep going. You have been my anchor, and I am eternally grateful for your presence in my life.*

To my amazing siblings, ***Akash, Athilesh, and Akash Verma*** *thank you for filling my life with laughter, comfort, and unconditional love. Whether through a word of encouragement or a shared smile, your presence has been a constant reminder that I am never alone.*

A truly special mention goes to ***Mishti and Kuku****, my adorable little bunnies, who brought joy to the darkest of days. Your playful innocence, cuddles, and boundless energy have been a source of emotional comfort, helping me find lightness and warmth when I needed it most. You may be small in size, but your love is monumental.* ***Dr. Trisha Verma, Rushikesh Kaknale, Dr. Mahitosh Lakhe, and Dr. Manisha Yersong*** *for their camaraderie, support, and willingness to lend a helping hand whenever needed. Our shared journey has made this*

experience all the more meaningful.

My heartfelt thanks to my respected seniors ***Dr. Vishwas, Dr. Hunny,*** *and* ***Dr. Shreya*** *whose guidance, encouragement, and belief in me helped shape my academic and professional path. Your mentorship has been invaluable.*

To my juniors ***Dr. Aarushi, Dr. Ishita, Dr. Unnati, Dr. Asila,*** *and* ***Dr. Niranjan*** *thank you for your enthusiasm, encouragement, and constant motivation. Your kindness and support have made this journey lighter, warmer, and truly enjoyable.*

In closing, this dissertation is a reflection of the collective strength, wisdom, and love of the people who have surrounded me. To each one of you, thank you for being a part of my journey, for your generosity of spirit, and for believing in me.

Dr. Ambika Verma

LIST OF FIGURES

SR NO.	TITLE	PAGE NO.
1	The Series Of Steps Involved In Carrying Out Finite Element Study	23
2	Cross Section Image Of Implant TMA Assembly	29
3(a)	Represents the stimulated 3D models of maxilla with implant placed at 17° with 17°TMA	29
3(b)	Represents the stimulated 3D models of maxilla implant placed at 17° with 30°TMA	30
3(c)	Represents the stimulated 3D models of maxilla with implant placed at 25° with 17°TMA	31
3(d)	Represents the stimulated 3D models of maxilla with implant placed at 25° with 30°TMA	31
3(e)	Represents the stimulated 3D models of maxilla with implant placed at 30° with 17°TMA	32
3(f)	Represents the stimulated 3D models of maxilla with implant placed at 30° with 30°TMA.	32
3(g)	Represents the stimulated 3D models of maxilla with implant placed at 35° with 17°TMA	33
3(h)	Represents the stimulated 3D models of maxilla with implant placed at 35° with 30°TMA	33
4(a)	Represents the stimulated 3D models of maxilla with implant placed at 17° with 17°TMA.	34
4(b)	Represents the stimulated 3D models of maxilla implant placed at 17° with 30°TMA	34
4(c)	Represents the stimulated 3D models of maxilla with implant	35

	placed at 25° with 17°TMA	
4(d)	Represents the stimulated 3D models of maxilla with implant placed at 25° with 30°TMA	35
4(e)	Represents the stimulated 3D models of maxilla with implant placed at 30° with 17°TMA	36
4(f)	Represents the stimulated 3D models of maxilla with implant placed at 30° with 30°TMA	36
4(g)	Represents the stimulated 3D models of maxilla with implant placed at 35° with 17°TMA	37
4(h)	Represents the stimulated 3D models of maxilla with implant placed at 35° with 30°TMA	37
5	Represents the dynamic loading of maxilla (direction of force and magnitude applied for all models was same)	38
6(a)	Represents stress transmitted on cortical bone- MODEL-1	61
6(b)	Represents stress transmitted on cancellous bone- MODEL-1	61
6(c)	Represents stress transmitted on abutment- MODEL-1	62
6(d)	Represents stress transmitted on implant- MODEL-1	62
7(a)	Represents stress transmitted on cortical bone- MODEL-2	63
7(b)	Represents stress transmitted on cancellous bone- MODEL-2	63
7(c)	Represents stress transmitted on abutment- MODEL-2	64
7(d)	Represents stress transmitted on implant- MODEL-2	64
8(a)	Represents stress transmitted on cortical bone- MODEL-3	65
8(b)	Represents stress transmitted on cancellous bone- MODEL-3	65
8(c)	Represents stress transmitted on abutment- MODEL-3	66

8(d)	Represents stress transmitted on implant- MODEL-3	66
9(a)	Represents stress transmitted on cortical bone- MODEL-4	67
9(b)	Represents stress transmitted on cancellous bone- MODEL-4	67
9(c)	Represents stress transmitted on abutment- MODEL-4	68
9(d)	Represents stress transmitted on implant- MODEL-4	68
10(a)	Represents stress transmitted on cortical bone- MODEL-5	69
10(b)	Represents stress transmitted on cancellous bone- MODEL-5	69
10(c)	Represents stress transmitted on abutment- MODEL-5	70
10(d)	Represents stress transmitted on implant- MODEL-5	70
11(a)	Represents stress transmitted on cortical bone- MODEL-6	71
11(b)	Represents stress transmitted on cancellous bone- MODEL-6	71
11(c)	Represents stress transmitted on abutment- MODEL-6	72
11(d)	Represents stress transmitted on implant- MODEL-6	72
12(a)	Represents stress transmitted on cortical bone- MODEL-7	73
12(b)	Represents stress transmitted on cancellous bone- MODEL-7	73
12(c)	Represents stress transmitted on abutment- MODEL-7	74
12(d)	Represents stress transmitted on implant- MODEL-7	74
13(a)	Represents stress transmitted on cortical bone- MODEL-8	75
13(b)	Represents stress transmitted on cancellous bone- MODEL-8	75
13(c)	Represents stress transmitted on abutment- MODEL-8	76
13(d)	Represents stress transmitted on implant- MODEL-8	76

LIST OF TABLES

SR NO.	TITLE	PAGE NO.
1	Number Of Nodes And Elements Forming Different Parts Of The Models	26
2	Mechanical Properties Of Different Materials	27
3	Showing Stress Transmitted on different parameters in Model-1	39
4	Showing Stress Transmitted on different parameters in Model-2	42
5	Showing Stress Transmitted on different parameters in Model-3	45
6	Showing Stress Transmitted on different parameters in Model-4	48
7	Showing Stress Transmitted on different parameters in Model-5	50
8	Showing Stress Transmitted on different parameters in Model-6	52
9	Showing Stress Transmitted on different parameters in Model-7	55
10	Showing Stress Transmitted on different parameters in Model-8	57
11	Combined Stress Analysis Table	60

LIST OF ANNEXURES

Sr.No.	Title	Page Number
1.	Ethical Clearance Certificate	110
2.	Anti-Plagiarism Certificate	111
3.	FEA Collaboration Certificate	112

LIST OF ABBREVIATIONS

SR NO.	ABBREVATIONS	FULL FORM
1.	CBCT	Cone Beam Computed Tomography
2.	CAD	Computer-Assisted Design
3.	FEA	Finite element analysis
4.	FEM	Finite Element Method
5.	ISP	Implant-Supported Prosthesis
6.	MPa	Mega Pascals
7.	mm	millimeters
8.	N	Newton
9.	SEM	Scanning Electron Microscope
10.	ANSYS R17	Software Used
11.	TMA	Transmucosal Abutment
12.	1st	First
13.	2nd	Second

14.	3D	Three-Dimensional
15.	°	Degrees
16.	%	Percentage

TABLE OF CONTENTS

SR. NO.	TABLE OF CONTENTS	PAGE NO.
1	INTRODUCTION	1-7
2	AIM & OBJECTIVES	8
3	RESEARCH HYPOTHESIS	9-10
4	REVIEW OF LITERATURE	11-22
5	MATERIALS & METHODOLOGY	23-38
6	RESULTS & OBSERVATIONS	39-76
7	DISCUSSION	77-92
8	LIMITATIONS	93
9	FUTURE SCOPE	94
10	SUMMARY	95-98
11	CONCLUSION	99-100
12	BIBLIOGRAPHY	101-109
13	ANNEXURES	110-112

Introduction

INTRODUCTION

Edentulousness represents more than just impaired oral function and bone loss; it often coincides with reduced self-confidence. The prevalence of edentulism continues to increase due to aging and the growing population of older adults[(1)]. Societies with comprehensive access to dental clinics tend to have lower rates of edentulism, while those lacking access experience higher rates. Despite advances in both curative and preventive dental care in recent decades, edentulism remains a persistent challenge for healthcare providers[(2).]

In the field of dental restoration for the edentulous jaw, a variety of prosthetic treatment options emerge, spanning from complete dentures to removable and fixed implant-retained prostheses. The integration of dental implants into fixed dental treatments marks a turning point in modern dentistry, significantly expanding treatment possibilities and improving patient satisfaction. This advancement not only broadens the options available but also delivers a more enduring and stable solution for addressing missing teeth [(3).]. Today, implant-supported full-arch restorations have risen to a standard and highly recommended treatment for individuals with fully edentulous arches [(3).].

Often referred to as the "third dentition," implant-retained restoration methods in rehabilitation closely mimic the original dentition, resulting in improved mastication and overall function. When compared to traditional complete dentures, implant-supported restorations can more effectively improve oral function and aesthetics in edentulous patients. Implant-supported restorations have shown satisfactory clinical outcomes and long-term success rates as the implants are successful in maintaining bone height and bone width i.e., preventing ridge resorption to quite an extent, as long as the implants remain healthy [(3).].

Contemporary dentistry aims to bring patients back to a state of normalcy in terms of profile, functionality, comfort, aesthetics, speech, and overall health, regardless of any atrophy,

disease, or injury affecting the oral and maxillofacial system. With people living longer on average, coupled with a substantial population already experiencing various dental issues, the field of implant dentistry is poised to remain vital for many generations of dental practitioners [(4)].

Fixed prosthesis provides improved stability, retention, phonetics, chewing efficiency, and comfort. Overall, they significantly improve the quality of life for patients, offering a more reliable, comfortable, and functional solution compared to removable dentures [(5).]. With chewing food being the primary function of oral prostheses, rehabilitation with a removable partial denture prosthesis does improve the ability to reduce bolus particle size and does not fully restore masticatory function.

A study conducted by **Soni R et al. (2020)** concludes that biting force and chewing efficiency improves with implant-supported over-dentures compared to conventional complete dentures [(7).]. So, implant-supported prosthetic rehabilitation enhances oral function, leading to an improved and more satisfying quality of life.

Anatomical constraints, such as buccal bone concavities and limited bone width in the maxillary anterior region, pose challenges for achieving optimal three-dimensional implant placement for aesthetic results. Often, implants must be placed at an angle due to anatomical variations, bone resorption patterns, or the presence of vital structures. However, adhering to the principles of fixed prosthodontics, these angulated implants need to be corrected to ensure successful restoration and long-term prosthesis durability. The use of angled the increasing preference for fixed prosthetic solutions to restore the edentulous maxilla has significantly driven the adoption of implant-supported prostheses in contemporary dentistry.

Angled abutments allow implants placed at steep angles to achieve a geometry similar to that of straight implants. Numerous studies have investigated the mechanical and biological properties of angled abutments, providing clinical data to validate their use. These studies indicate that angled abutments produce outcomes comparable to those of straight abutments, with no significant clinical differences observed.

Finite element analysis (FEA) facilitates detailed visualization of structural behaviours, such as bending or twisting, and maps the distribution of stresses and displacements. Additionally, FEA offers versatile simulation capabilities to manage the complexity of modelling and analysing systems, making it a valuable tool for studying the maxilla.

The three-dimensional Finite element method has been very useful in studying the biomechanics of stress on dental implants, bone, along with that it also helps us to evaluate the fatigue behavior of the implants and the prosthetic components In a study by **M. Taruna et al explained** that tilting of the posterior implants makes it possible to achieve good anchorage in bone and also does not interfere with the mental foramen in severely resorbed cases, gives us a correct understanding as to why tilting of posterior implants is a viable treatment protocol[(10).].

Bevilacqua et al. reported that angulating the distal implant at 30° in a fixed prosthesis significantly reduced stress levels—by approximately 52% in cortical bone and 47.6% in cancellous bone—when compared to vertically placed implants [(11).].

In a study by **Aaron Yu-Jen Wu et al** changing the position of anterior implants has only a minor effect on the stress and strain in the surrounding bone, so we are only tilting the distal implants and not the mesial ones [(12).].

The All-on-4 concept, introduced by **Paulo Malo**, presents an effective treatment strategy by utilizing a mix of straight and angled multi-unit abutments [13]. This method enables immediate loading of a full-arch prosthesis supported by just four implants, offering a dependable and efficient solution for edentulous patients.

Multi-unit abutments offer significant versatility by correcting implant angulation and aiding in soft tissue contouring. They simplify the impression-taking process and broaden prosthetic possibilities. Since their introduction in 2000, multi-unit abutments have transformed dental practice, setting a standard that has inspired numerous similar designs.

Despite the success of Osseo integrated implants, long-term observations reveal various biological and mechanical complications. Although dental implants are innovative, they remain susceptible to failures, particularly in the screw connections of the implant system. Nonetheless, advancements in oral rehabilitation continue to progress, fuelled by innovation and a commitment to enhancing Oral Health-Related Quality of Life for patients with partial or complete tooth loss[14].

Angled abutments often face off-axis loading, leading to uneven force distribution that may compromise their performance, particularly affecting the thin labial cortical plate. The success of an implant depends significantly on the amount and pattern of stress transferred to the surrounding bone. While moderate stress can promote bone remodelling, excessive or concentrated forces from angled abutments may cause bone resorption or threaten osseointegration.

The introduction of the All-on-4 treatment concept, where distal implants can be tilted up to 45°, has increased the use of multi-unit abutments. Angled multi-unit abutments, available at fit, and reliable function, significantly improving the efficacy of implant-supported

restorations. In biomechanical evaluations of dental implants, the finite element method (FEM) is widely used due to its ability to simulate complex clinical conditions more effectively than other analytical approaches. In the maxillary anterior region, the palatal surfaces of upper teeth guide the lower anterior teeth during protrusive and lateral movements, regardless of the occlusal scheme, often resulting in angulated forces on implants that can lead to issues like screw loosening[16].

Due to changes in bone morphology, implants in the anterior maxilla often require increased angulation to meet spatial and aesthetic requirements. Angled abutments provide a practical solution in such cases. Poorly designed implants can create areas of high stress in the surrounding bone, potentially leading to significant resorption, progressive loosening, or implant failure.

Therefore, analysing stress distribution across various implant systems is critical for optimizing implant design. Stress and strain assessments have utilized methods such as photoelasticity, strain gauge analysis, and finite element analysis[17]. Photoelasticity provides useful qualitative insights into stress distribution but lacks precision for quantitative measurements. Strain gauges offer accurate strain data but only at specific points. In contrast, finite element analysis allows precise quantitative evaluation of stress at any point within a mathematical model, with both 2D and 3D FEM used to study stress distribution around different implant systems.

Most prior studies have focused on how implant design affects stress distribution in surrounding bone and the mechanical factors contributing to abutment loosening. However, since abutment shape significantly influences force transmission within the implant system, this study aims to evaluate how two different abutment designs affect stress patterns in the under two loading conditions: purely vertical force and a combination of vertical and 45° oblique loading[18].

The 3D finite element method, a numerical simulation technique used in engineering, assesses stress, strain, and displacement in structures under internal or external forces. It employs a geometrically accurate virtual model composed of numerous discrete elements connected at nodal points, offering the benefit of being entirely non-invasive. Limited research exists on stress evaluation in tilted dental implants with varying angulations of transmucosal abutments, prompting the need for this study.

When dental implants are positioned at an angle, they often require angled abutments to correct the alignment, which can significantly amplify tangential forces. According to research, a 10° increase in the cusp inclination of a prosthesis can lead to roughly a 30% rise in torque, whereas a 10° increase in implant angulation results in only about a 5% torque increase. The angle of the prosthesis has a greater impact on force distribution than the implant's angle, as excessive forces may compromise osseointegration. To address offset loading, corrective solutions like angled abutments are advised. More recently, transmucosal abutments have been utilized to minimize offset forces and better align them with the implant's central axis.

Although studies have investigated stress on prosthetic screws and surrounding bone, there is limited research comparing how implant angulation affects stress distribution or evaluating stress patterns from angled transmucosal abutments used to correct tilted implants. While some data exists on transmucosal abutments, few studies have explored how their angulation interacts with the angulation of the underlying tilted implant in terms of stress on the implant and surrounding bone in a simulated maxilla. This gap highlights the need for further investigation.

This study will use the 3D finite element method to assess how transmucosal abutment angulation influences stress distribution in and around tilted dental implants in the maxillary arch restored with multi-unit abutments.

Aim & Objectives

AIM

To evaluate the effect of implant tilt and angulation of transmucosal abutments on generation of stress in an around tilted dental implant in stimulated maxillary arch using Finite Element Analysis.

OBJECTIVES

a) To evaluate the stress generated in and around tilted dental implant in simulated maxillary arch at 17°, 25°, 30°, 35° restored with 17° multiunit abutment.

b) To evaluate the stress generated in and around tilted dental implant in simulated maxillary arch at 17°, 25°, 30°, 35°and restored with 30° multiunit abutment.

c) Comparative analysis of the stress generated in tilted dental implant in simulated maxillary arch at different angulations and restored with 17° vs 30° multiunit abutment.

Research Hypothesis

NULL HYPOTHESIS

There is no difference in the stress transmission in and around tilted dental implants at an angulation of 17^0 , 25^0 , 30^0 ,35^0 restored with transmucosal abutments with 17^0 and 30^0

ALTERNATIVE HYPOTHESIS

There is a difference in the stress transmission in and around tilted dental implants at an angulation of 17^0 , 25^0 , 30^0 ,35^0 restored with transmucosal abutments with 17^0 and 30^0.

NEED OF THE STUDY

Whenever there is an offset in implant angulation it has to be restored with angulated abutment. Consequently, the forces generated due to resultant tangential force increases manifold. Weinberg in his article has given that, a 10° increase in cusp inclination will produce approximately a 30% increase in torque compared with only 5% increase in torque with a 10° increase in implant inclination. The angle of the resultant prosthesis if more important than the implant itself. These forces tend to be very harmful to the success of osseointegration. Corrective measures in form of angulated abutments are recommended to reduce the offset loading. More recently transmucosal abutments have been used to reduce offset loading and direct the forces closer to the midline of implant head. A variety of literature has been published evaluating the stress on prosthetic screw, peri implant bone but none of those studies compared the effect of implant angulation on stress and also evaluate the stress patterns developed as a result of the use of angulated transmucosal abutment used for correction of angulated implants. Various studies have been done on transmucosal abutment but not enough studies have been

found to evaluate the difference between the angulation of the transmucosal abutment and the underlying angulated implant in terms of stress generated in and around the tilted implant in the simulated maxilla hence, this study has been planned. Hence, magnitude of angulation as corrected improved by transmucosal abutment and its impact on the stress distribution along the implant body will be assessed.

Review Of Literature

REVIEW OF LITERATURE

L A Weinberg , B Kruger(1994) in a study observed the force distribution of multiple tooth-supported and implant-supported prostheses is completely different. A direct correlation exists between the degree of flexion at the site of loading and the amount of force distribution to other members of the prosthesis. Micromovement produced by the periodontal fibers facilitates force distribution to all the root surfaces of the natural tooth abutments. The rigidity of the implant/abutment/prosthesis configuration concentrates the force at the crestal bone at the site of loading with limited distribution to the remaining implants. Differential mobility concentrates the force distribution to the bone support of the most rigid members of splinted natural teeth or to the implants when they are united with natural teeth in a combined prosthesis. Implants always support the natural teeth and never the other way around. Therefore, a nonrigid attachment is recommended between a tooth-supported prosthesis and an implant-supported prosthesis when they are combined. However, when implants are interspersed with natural teeth in the same prosthesis, the restoration will be implant borne. This requires special force distribution analysis to prevent implant overload.

L A Weinberg(1998) evaluated through a study that all physiologic biomechanical processes are interrelated and, therefore, reactive, which produces an accumulative effect that can cause implant overload. A new approach called "therapeutic biomechanics" suggests using corrective procedures to reduce implant loading. The head of the implant is positioned as close to the midline of the restoration as possible. Implant inclination may be required, but it produces less torque (moment) than horizontal implant offset. Posterior cross occlusion should be used where possible to decrease horizontal implant offset. Angulated, or reangulated abutments provide access or parallelism as needed. The posterior cusp inclination should be markedly reduced. When a vertical overlap is present anteriorly, a horizontal stop on the maxillary lingual surface

redirects harmful lateral force to be vertical toward the implant and supporting bone. Because of physiologic variability, creating a modified centric occlusion that contains a 1.5-mm fossa will produce vertical resultant force within this range of motion rather than lateral force.

Geng JP, Tan KB et al (2001) conducted a study on application of finite element analysis in implant dentistry. Finite element analysis (FEA) has been used extensively to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on implant success. By understanding the basic 11 theory, method, application, and limitations of FEA in implant dentistry, the clinician will be better equipped to interpret results of FEA studies and extrapolate these results to clinical situations. This article reviews the current status of FEA applications in implant dentistry and discusses findings from FEA studies in relation to the bone–implant interface, the implant-prosthesis connection, and multiple-implant prostheses. Alkan and co-workers (2004)22 designed a study to investigate stress distribution of preloaded dental implant screws in 3 implant-to-abutment joint systems, viz (1) Branemark external hexagonal screw-retained abutment, (2) ITI 8-degree morse tapered cemented abutment, & (3) ITI 8-degree Morse tapereed plus internal octagonal screw retained abutment, under simulated occlusal forces using 3D finite element analysis. They simulated the preload resulting from the manufacturers' recommended torques in implant screw joint assemblies using a thermal load and contact analysis method and then these preloaded implants were loaded with 3 simulated static occlusal loads (10N; horizontal, 35N; vertical, 70N; oblique) on the crown position onto the implant complex. They observed that stresses increased in both the abutment and prosthetic screws after simulated horizontal loading, however, when vertical and oblique static loads are were, they observed that the stresses decreased in the external hexagonal and internal octagonal plus 8-degree morse tapered abutment & prosthetic screws, with exception of prosthetic screw of ITI abutment after 70N oblique loading. Also, stresses were observed to increase in The ITI - degree Morse tapered cemented abutment after

application of oblique and vertical loads. They finally concluded that within the limitation of this finite element study, the 3 abutment-to-implant joint systems tested did not demonstrate stress values that would cause screw failure or loosening for single tooth restorations.

Heoung-Jae Chun et al (2006) did a study to investigate the effect of 3 different abutment types on the stress distribution in bone with inclined loads using finite element analysis. The 1-body, internal-hex, and external-hex implant systems were modelled to study the effect of abutment type on stress distribution in bone. The bone model used in this study comprised compact and spongious bone assumed to be homogeneous, isotropic, and linearly elastic. : It was found that the internal-hex implant system generated the lowest maximum Von Mises stresses for all loading conditions because of reduction of the bending effect by sliding in the tapered joints between the implant and abutment. t was concluded that abutment type has significant influence on the stress distribution in bone because of different load transfer mechanisms and the differences in size of the contact area between the abutment and implant.

Silva, Mendonca, Lopes et al (2009) conducted a study to compare the biomechanical behavior of all-on4 system with that of a six-implant-supported maxillary prosthesis with tilted distal implants by developing three-dimensional models representing maxillae restored with an all-on-4 and with a six-implant-supported prosthesis, in a three-dimensional design software and then transferring into FEM software. They subjected the models to four different loading simulations viz, full mouth biting, canine disclusion, load on a cantilever and load in the absence of cantilever and localized and quantified maximum von Mises stresses for comparison. They observed the peak stress points to be located on the neck of distal tilted implant in both models, in all loading simulations and also that von Mises stress values were higher in ‘all-on-4’ model (7% to 29% higher) and in presence of cantilever the von Mises stress valuess increased by about 100% in both models. They finally concluded that addition

of implants results in reduction of maximum von mIses stress values and cantilever greatly increased the stress.

P Cardelli et al (2009) conducted a study to assess the stress distribution on the bone and on the prosthetic components was evaluated by means of a Finite Element Analysis (FEA). The effect of abutment axis angle was investigated considering three different measures: 15°, 25° and 35°. The simulation of both 450N maximal biting and 100N functional forces, highlighted the stress trend, which suggests as a maximum limit to choose an abutment with an angulation of 25°. Results of this study justify some clinical and radiographic evidence, in which bone resorption was recorded at the level of the implant neck in close contact with the cortical bone. As far as angulation is considered we concluded that, if it is necessary to use them, it is suggested to not exceed the limit of 25° and if possible, to apply them exclusively in the antetior region, where the forces involved are significantly lower, or with another implant.

Siddharth Shelat et al (2011) conducted a study to investigate the effect of two different abutment types on stress distribution in the bone around an implant under two loading conditions, vertical load and combined load (vertical + angle of 45°). Implant of 4.2 × 12mm2 was used. Two 2-piece implant systems, Internal Hex and External Hex implant-abutment complex were used. The implant-abutment complex was embedded in bone and subjected to static load of 100 N vertically and a combined load (vertical + 45° angulation). Finite Element modeling of bone implant and abutment was done using Ansys Classical 10.0 software. They observed maximum Von Mises stress occurred in the region of the compact bone under all loading conditions irrespective of the type of abutment use. Significant reduction in Von Mises stress was observed at the boundary between compact and spongious bone because of relatively low elastic modulus of spongious bone. 13 Saber and co-workers (2015)25 conducted a study to evaluate the amount and distribution of stress in maxillary bone surrounding the implants with all-on-four vs frequently used method with six implants technique using different numbers

and inclination angles, using finite element analysis, wherein, they created a five-3D edentulous maxillary models and virtually placed implants anterior to the maxillary sinus and splinted it with a superstructure. In 1st to 4th models, they placed 4 implants with distal inclination of 0, 15, 30 and 45 degrees, respectively. In the 5th model they placed six virtual implants and a load of 100N was placed in the left most distal region of the superstructure and they evaluated the maximum von Mises stress values in cancellous and cortical bone. The maximum stress values in cancellous bone and cortical bone obseved by them were 7.15 Mpa& 51.69 Mpa, respectively (Model I) and reduction in stress values in Models II toV were observed as 6%, 18%, 54% and 24% in cancellous bone and 12%, 36%, 62% and 62% in cortical bone, respectively. They finally concluded that increasing the inclination in posterior implants resulted in reduction in cantilever length and maximum stress decline in both cancellous& cortical bone.

Shruti Khaneja et al (2016) conducted a study to investigate if the use of tilted implants as distal abutments is biomechanically superior to the use of distal cantilevers implants as distal abutments is biomechanically superior to the use of distal. A 3-D edentulous maxillary model was generated computed tomography images. Four implants were placed in the pre-m and splinted with a titanium bar. Keeping the prosthesis length co six different configurations were evaluated with the distal implant var length (8 mm, 10 mm, 13 mm); angulation (0, 15, 30, 45°) and abu angulation (0, 17, 30°). A vertical load (120 N) was applied on the tit bar corresponding to the position of the left second premolar abu Von Mises stress values transmitted to the peri-implant bone and im abutment junction were evaluated. The maximum stresses recorded at the peri-implant bon the implant abutment junction was 5.821 MPa and 8.261 MPa for vertical implant; 5.358 MPa and 8.131 MPa for the 10 mm implan 4.897 MPa and 7.754 MPa for the 13 mm vertical implant. Tilted implants resulted in increased stress values for all variables: 8.906 8.938 MPa and 8.994 MPa at the peri-implant bone and 14.636 15.567 MPa and 16.872

MPa at the implant abutment junction for and 45° respectively. An increase in implant length resulted in a decrease stresses, whereas the tilted distal implants and abutments increas stresses at the periimplant bone and implant abutment junction implants.

Eduardo Borie et al (2016) conducted a study to to analyze the influence of three different transmucosal heights of the abutments in single and multiple implant-supported prostheses through the finite element method. External hexagon implants, MicroUnit, and EsthetiCone abutments were scanned and placed in an edentulous maxillary model obtained from a tomography database. The simulations were divided into two groups: (1) one implant with 3.75 × 10 mm placed in the upper central incisor, simulating a single implant- 14 supported fixed prosthesis with an EsthetiCone abutment; and (2) two implants with 3.75 × 10 mm placed in the upper lateral incisors with MicroUnit abutments, simulating a multiple implant-supported prosthesis. Subsequently, each group was subdivided into three models according to the transmucosal height (1, 2, and 3 mm). A static oblique load at an angle of 45 degrees to the long axis of the implant in palatal-buccal directioatal-buccal direction of 150 and 75 N was applied for multiple and single implant-supported prosthesis, respectively. The implants and abutments were assessed according to the equivalent Von Mises stress analyses while the bone and ceramics were analyzed through maximum and minimum principal stresses. The total deformation values increased in all models, while the transmucosal height was augmented. The transmucosal height of the abutments influences the stress values at the bone, ceramics, implants, and abutments of both the single and multiple implant-supported prostheses, with the transmucosal height of 1 mm showing the lowest stress values. Moustafa and co-workers (2018)24 conducted a study to determine whether 1-piece and 2-piece narrow diameter implants with equivalent geometries exhibit stresses and strains differently under applied loading conditions when used to retain a mandibular overdenture and to evaluate how stresses are transmitted to the surrounding bone, using 3D finite element analysis, wherein, they

structured a computer based numerical model for the anterior segment of the mandible with, 2 narrow diameter implants retaining a mandibular overdenture, one being a 1-piece and the other being a 2-piece; applied 35N & 100N loads through the overdenture and analysed the Von Mises stresses along the implants and surrounding bone. They observed that stresses around the2-piece design were greater than those around the 1-piece design. Thus they finally concluded that a 2-piece narrow diameter implant can be a reliable treatment option to retain overdentures in cases where immediate loading is not recommended. Ahmed et al (2019)3 conducted a study to comparatively evaluate the effect of cyclic loading on abutment screw loosening of 25° angled premachined titanium abutments with that of 25° angled custom cast cobalt chromium (Co-Cr) abutments. Mean pre-cyclic loading reverse torque values were 28.96 Ncm (Group I) and 26.01 Ncm (Group II) and mean postcyclic loading reverse torque values were 25.84 Ncm (Group I) and 23.82 Ncm (Group II) respectively. Cyclic loading resulted in significantly lesser reverse torque values within both test groups ($p<0.01$). There was significantly greater loss of preload for Group II samples both prior to and after cyclic loading compared to Group I samples ($p<0.01$). Conclusion was Cyclic loading resulted in significant screw loosening within both test groups. Screw loosening was found to be significantly less for angled premachined titanium abutments than for angled custom cast Co-Cr abutments both before and after cyclic loading. The study is clinically significant regarding the importance of screw joint-abutment connection 15 acting in a maxillary anterior implant region under load and the consequences of poor connection. The study is important in the aspect of clinical implications of angled implant abutments and the effect of loading on them. García-Braz et al (2019)14 conducted a study to analyse the stress distribution on two types of extra-short dental implants with 5 mm of length: An internal hexagon (IH) and morse taper connection (MT). The three-dimensional model was composed of trabecular and cortical bone, a crown, an extra-short dental implant and their components. An axial load of 150 N was applied and another

inclined 30° with the same magnitude. Stress concentrations on the IH implant are observed in the region of the first threads for the screw. However, in the MT implant the highest stress occurs at the edges of the upper implant platform. In view of the results obtained in this study the two types of prosthetic fittings present a good stress distribution. The Morse taper connections presented better behavior than the internal in both loading configurations.

Ji- Hyeon Oh et al (2020) conducted a study to evaluate the stress patterns induced in the prosthetic screws by different prosthetic screw and abutment designs in all-on-4 concept using finite element analysis using 6 groups of each screw type, viz, short and narrow screw, short abutment; short and wide screw, short abutment; long and wide screw, short abutment; short and narrow screw, long abutment; short and wide screw, long abutment; and long and wide screw, long abutment. Each group was tested under a cantilever loading of 200N that was applied on the farther posterior to the position of the connection between the distal implant &the metal framework. They observed the stress values to decrease in the posterior prosthetic screws as the length and diameter increased. They finally concluded that the long and wide screw design offers advantages in stress distribution when compared with short and narrow design.

Udita Thakkar et al (2020) did a study to evaluate the stress distribution around the mini-implant during maxillary anterior intrusion under different conditions of different angulations and different positions of implant. Finite element analysis was carried out. Stress under the following 4 conditions was analyzed: (a) single central implant placed at 90°, (b) single central implant placed at 120°, (c) bilaterally placed implant at 90°, and (d) bilaterally placed implant at 120°. The displacement seen with 90° angulation in the single implant case is less compared with the 120° angulation case for all the 6 maxillary anterior teeth. Also, in the bilateral implant case, the Von Mises stress is less when the 90° angulation case is compared to 120° angulation case. But in bilaterally placed implant, the stress gets distributed evenly in the anterior region.

The stress in 90° angulation cases seems to be concentrated at the center. Stresses measured on the teeth are less and distributed 16 more evenly when the point of force application is bilateral. It was also observed that the stress increases with increase in the angulation of the implant. As the contact between the implant and the bone increases, the stability increases. Hence, the implant should be obliquely inserted into the bone. Concentrated stresses are not favourable as they can increase the risk of bone and root resorption.

Hegde and co-workers (2020) evaluated a study to compare the biomechanical behaviour of the twoimplant supported over denture design with the all-on-four concept and to eveluate the Von Mises stresses induced on the implant under different loading simulations using the 3D finite element method. They compared 3D models representing mandible restored with two implant supported prosthesis with the models that were restored based on the all-on-four concept by subjecting each model to 4 different loading simulations (full mouth biting, canine disclusion, load on cantilever and load in the absence of cantilever. They observed that under all loading simulations, the maximum stress concentrations were along the neck of the implant and the stress levels for the full mouth loading simulation was highest for two-implant supported overdenture design and the least for all-onfour overdenture design. They finally concluded that the all-on-four concept is a clinically applicable treatment option for the atrophic edentulous ridges and induces least number of stresses on the edentulous ridges.

G Deste and R Durkan (2020) conducted a study to observe the stresses of all-on-four implant designs in an edentulous mandible in the implant, surrounding bone and monolithic ceramics using four different mandibular all-on-four implant models viz, 1M, 2M, 3M in which they placed anterior implants vertically and posterior implants were inclined at 15º with a cantilever of length 5mm(1M); 15º with a cantilever of length 9mm (2M); 30º with a cantilever of length 5mm(3M) and 30º with a cantilever of length 9mm (4M). They made the full arch-arch fixed prosthesis in monolithic zirconia framework and monolithic sodium disilicate superstructure

and calculated the stress values after applying 300N bilateral force at an angle of 30º and oblique to the occlusal plane. They observed highest stress concentration in the 2nd connector region between the canine & 1st premolar tooth in the monolithic zirconia framework (457.21 Mpa); in cortical bone 60.93 Mpa stress in posterior implants; higher stress in posterior angled than straight implants; increase of stress by 21 Mpa in implants angled at 15º. They finally concluded that in bilateral loading, force applied to anterior implants does not have a significant effect on bone structure but stress concentration increases in posterior angled implants & surrounding bone, also stress concentration level and localization are affected by increasing the posterior implant angulation.

Sumedha Kapoor et al (2021) conducted a study to assess and correlate the stress distribution in an anterior maxillary implant-supported prosthesis with 0°(degree), 15°, and 25° angulated titanium and zirconia abutments using a three-dimensional (3D) finite element analysis (FEA). Within the limitations of this study, straight abutments generated a more uniform and minimal stress in implant and peri-implant bone than angulated abutments. Titanium abutments generated higher stress levels than zirconia abutments. The stresses generated are directly proportional to an increase in abutment angulation, and therefore, straight abutments are most suitable for favourable stress transmission.

Paramba Hitendrabhai Acharya et al (2021) evaluated a study to assess peri-implant stress distribution using finite element analysis in implant supported fixed partial denture with occlusal schemes of cuspally loaded occlusion and implant protected occlusion. A 3-D finite element model of mandible with D2 bone with partially edentulism with unilateral distal extension was made. Two Ti alloy identical implants with 4.2 mm diameter and 10 mm length were placed in the mandibular second premolar and the mandibular second molar region and prosthesis was given with the mandibular first molar pontic. Vertical load of 100 N and and oblique load of 70 N was applied on occlusal surface of prosthesis. Group 1 was cuspally

loaded occlusion with total 8 contact points and Group 2 was implant protected occlusion with 3 contact points. In Group 1 for vertical load, maximum stress was generated over implant having 14.3552 Mpa. While for oblique load, overall stress generated was 28.0732 Mpa. In Group 2 for vertical load, maximum stress was generated over crown and overall stress was 16.7682 Mpa. But for oblique load, crown stress and overall stress was maximum 22.7561 Mpa. When Group 1 is compared to Group 2, harmful oblique load caused maximum overall stress 28.0732 Mpa in Group 1.

Özge Özdal Zincir et al (2021) conducted a study to compare and evaluate the stress and tension values in conventional angled multiunit abutment-implant connection systems and monoblock dental implants used in the all-on-four procedure with finite element analysis. Two master models were created by placing four implants connected to multiunit abutments (group A) in the interforaminal region of a completely edentulous mandible and four monoblock implants (group B) in the same region of another completely edentulous mandible. Group A implants were classified according to their diameter as follows: 3.5 mm (M1A), 4.0 mm (M2A), and 4.5 mm (M3A). Similarly, group B implants were classified as M1B, M2B, and M3B. In the six models rehabilitated with acrylic fixed prostheses, a 100 N force was applied to the anterior implant region, and a 250 N force was applied to the posterior cantilever in both axial and 30° oblique directions. Von Mises stresses were analyzed in the bone and implant regions of all models. M1A and M1B, M2A and M2B, and M3A and M3B were compared with each other under axial and oblique forces. The maximum Von Mises 18 stresses in the bone around implants and the prosthesis screws, and the maximum and minimum principal stresses in the cortical and trabecular bone in group A models were significantly higher than those in group B models.

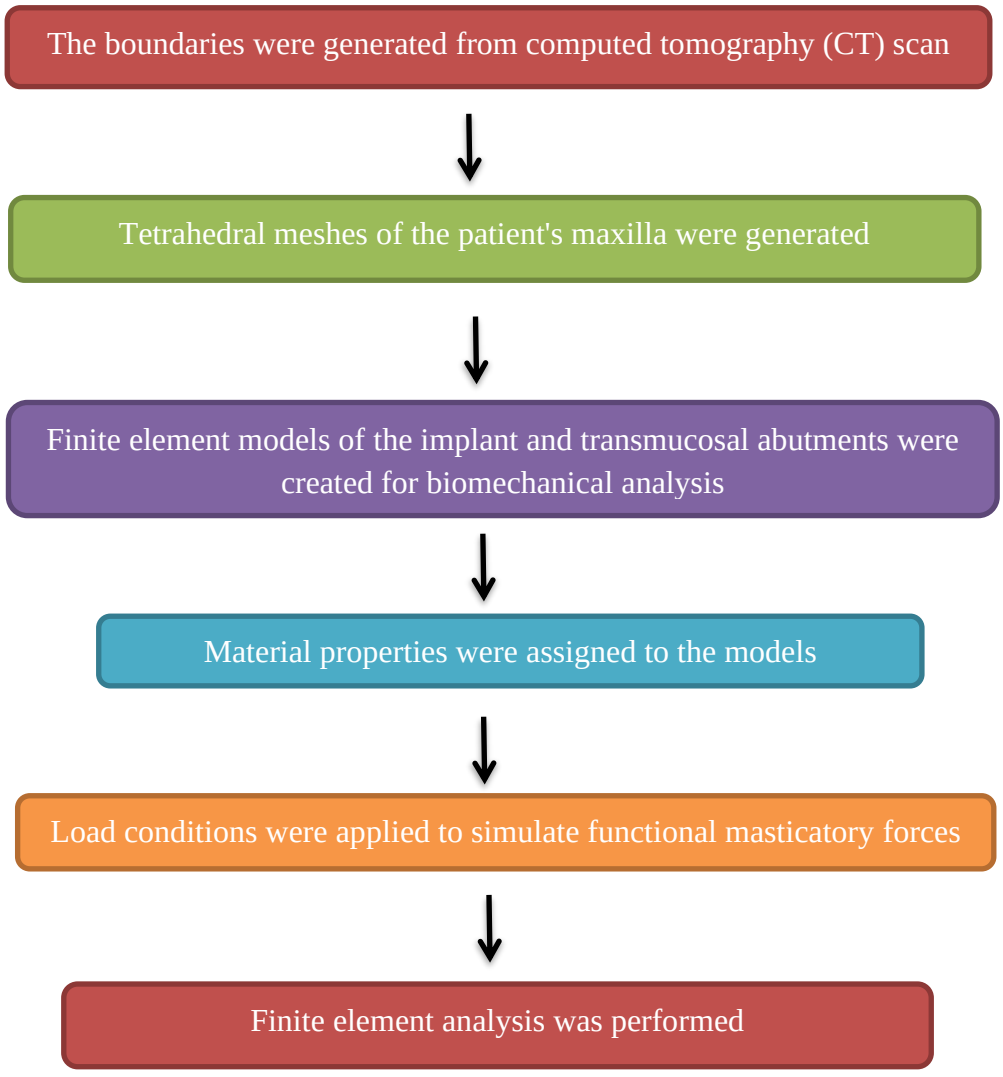
Dr.Akshay Bhargava et al (2022) conducted a study to evaluate the quantitative changes in crestal bone levels in palatally positioned two-piece endosseous implants in maxillary posterior region. The placement of implants in atrophic bone has always posed a challenge to the clinicians in terms of good surgical and esthetic results. The placement of implants palatal to the alveolar crest in maxilla allows maximum use of the available bone in patient with severe bone resorption. Among Group I (crestal bone level evaluated on mesial side using RVG), three months post implant placement showed the highest Crestal Bone Loss i.e. 0.66±0.070 mm and immediately post implant placement evaluation showed the least value i.e. 0.04±0.07 mm which was statistically significant (p-value < 0.05). Among Group II (crestal bone level evaluated on distal side using RVG), three months post implant placement evaluation showed the highest Crestal Bone Loss i.e. 0.66±0.070 mm and immediately post implant placement evaluation showed the least value i.e. 0.04±0.07 mm which was statistically significant (p-value < 0.05). The mean Crestal Bone loss at the end of 3 months post implant placement observed was 0.29mm and at the end of 6 months post implant placement observed was 0.66mm. The effect of tilting the implants palatally by 15 to 25 degrees did not considerably influence the Crestal Bone Levels around the implant.

Materials & Methodology

MATERIALS AND METHODOLOGY

The present in-vitro FEM study was conducted in the Department of Prosthodontics and Crown & Bridge, Santosh Dental College and Hospital, Delhi-NCR. Finite Element Analysis software used for performing the study was ANSYS R17

FIGURE 1- The Series of Steps Involved in Carrying out the Finite Element Study


The boundaries were generated from computed tomography (CT) scan
Tetrahedral meshes of the patient's maxilla were generated
Finite element models of the implant and transmucosal abutments were created for biomechanical analysis
Material properties were assigned to the models
Load conditions were applied to simulate functional masticatory forces
Finite element analysis was performed

Using ANSYS 17 for finite element analysis (FEA) provides several significant advantages. Its improved user interface simplifies the setup of simulations, visualization of results, and modification processes, thereby saving time and reducing errors. Additionally, ANSYS 17 offers more accurate simulation of material properties, leading to realistic predictions of implant behavior. The advanced meshing algorithms ensure higher quality and more refined meshes, enhancing the precision and reliability of the analysis.

A 3D model of the human maxilla was simulated from the Cone Beam Computed Tomographic (CBCT) scan data of a completely edentulous patient. The CBCT scan data was transformed into a solid model of the maxilla using modeling software, namely, SolidWorks 2014. Cortical bone was established around the cancellous core. Also, a 3D model of a titanium dental implant (Adin) of the size 3.75 x 11.5 mm at an angle of 17° ,25°, 30°, 35° and angulated (17° & 30°) Transmucosal abutments, were simulated using Solid Works 2014.

As in the model of single implant, one anterior implant was placed at 17°,25°,30°,35° angulation respectively. [(22)] to which the Transmucosal abutments 17° & 30°) were connected, was also simulated by using the same modeling software. [(28)].

Eight different models were simulated in which implant of the size 3.75mm x 11.5mm was placed in anterior region (central incisor) positions as described below which are represented in models. MODEL-1 - implant placed at 17° with TMA-17, MODEL-2 - implant placed at 17° with TMA-30°, MODEL-3 - implant placed at 25° with TMA-17, MODEL-4 - implant placed at 25° with TMA-30° , MODEL-5 - implant placed at 30° with TMA-17° MODEL-6 - implant placed at 30° with TMA-30° MODEL-7 - implant placed at 35° with TMA-17° MODEL-8- implant placed at 35° with TMA-30°.

The 8 isotropic and homogeneous 3d models were represented and shown in figures 3(a), 3(b), 3(c), 3(d), 3(e), 3(f), 3(g), 3(h). Figure 3a represents the stimulated 3d models of maxilla with implant placed at 17° with 17°TMA. Figure 3b represents the stimulated 3d models of maxilla implant placed at 17° with 30°TMA. Figure 3c represents the stimulated 3d models of maxilla with implant placed at 25° with 17°TMA. Figure 3d represents the stimulated 3d models of maxilla with implant placed at 25° with 30°TMA. Figure 3e represents the stimulated 3d models of maxilla with implant placed at 30° with 17°TMA. Figure 3f represents the stimulated 3d models of maxilla with implant placed at 30° with 30°TMA. Figure 3g represents the stimulated 3d models of maxilla with implant placed at 35° with 17°TMA. Figure 3h represents the stimulated 3d models of maxilla with implant placed at 35° with 30°TMA

The constructed 3D models were then exported into Finite Element software (ANSYS R17). These models were then meshed by tetrahedral elements. The meshing element type is tetrahedron nodal element. All these properties were included in Ansys software library before starting the simulation analysis. The Mesh models were generated and shown in figures 4(a), 4(b), 4(c), 4(d), 4(e), 4(f), 4(g), 4(h). Figure 4a represents the stimulated 3d models of maxilla with implant placed at 17° with 17°TMA. Figure 4b represents the stimulated 3d models of maxilla implant placed at 17° with 30°TMA. Figure 4c represents the stimulated 3d models of maxilla with implant placed at 25° with 17°TMA. Figure 4d represents the stimulated 3d models of maxilla with implant placed at 25° with 30°TMA. Figure 4e represents the stimulated 3d models of maxilla with implant placed at 30° with 17°TMA. Figure 4f represents the stimulated 3d models of maxilla with implant placed at 30° with 30°TMA. Figure 4g represents the stimulated 3d models of maxilla with implant placed at 35° with 17°TMA. Figure 4h represents the stimulated 3d models of maxilla with implant placed at 35° with 30°TMA.

The total number of elements and nodes in different parts of each meshed model are shown in Table 1. All implants and abutments were modeled based on the actual dimensions available from the commercial data sheets of each manufacturer.

Table 1 - Number of Nodes and Elements forming different parts of the models

Model	Total		Cortical		Cancellous		Implant		Abutment	
Implant-Abutment	Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes
Model 1 17^0-17^0	27659	48038	13671	23114	3918	7113	7569	13319	2501	4492
Model -2 17^0-30^0	26822	46448	13155	22219	3399	6097	7569	13319	2699	4813
Model -3 25^0 -17^0	25830	44864	13100	22173	2674	5004	7569	13319	2387	4368
Model-4 25^0 -30^0	25714	44820	12830	21811	2714	5042	7569	13319	2601	4648
Model-5 17^0-30^0	25932	44986	12958	21922	2551	4630	7569	13319	2854	5115
Model-6 30^0-30^0	27333	47193	12823	21655	3403	6085	7569	13319	3538	6134
Model-7 35^0-17^0	25746	44440	12702	21513	2102	3723	7569	13319	3373	5885
Model-8 35^0-30^0	26348	45935	12276	20935	2548	4796	7569	13319	3955	6885

The analysis software started by importing the models with all parts to an efficient 3D analysis software Ansys17.

The purpose of static analysis is to predict the behavior of a model in response to the loading conditions independent of time. The properties of the material in static analysis used are the Modulus of elasticity, the Poisson's coefficient, the density of the material that aids us to determine the Maximum and Minimum stress of the particular area of interest.

Then, the materials used for each part were selected, taking into account the specifications and determination of their properties with great precision. The material properties that are the mechanical properties of the materials used were defined based on the type of analysis. In this study, dynamic analyses were performed.

Table 2 – Mechanical Properties Different Materials [29][30]

	YOUNG`S MODULUS	POISSON`S RATIO	TENSILE STRENGTH	DENSITY
CORTICAL BONE	13700MPa	0.30	114MPa	1990kg/m^3
CANCELLOUS BONE	1370MPa	0.30	52MPa	1847kg/m^3
TITANIUM ALLOY	110000MPa	0.35	834MPa	4400kg/m^3

Boundary conditions were delineated, particularly at the peripheral nodes of the bone. The movements of the nodes were constrained in this area. The bone-implant interface was considered completely fixed and the implants were considered to be Osseo-integrated to the maximum extent. A perfect fit situation was assumed between the implants, bone, and the prosthetic structure. The necessary load conditions were applied to the model parts at the selected positions according to the study requirements.

To carry out loading, all the models were subjected to a load of 200N. shown in figure 7. Static structural analysis was conducted in ANSYS R17.0, and the following conditions were applied:

Fixed Support (A): The superior surface of the maxillary model was constrained to simulate fixation of the bone and surrounding structures, preventing any movement during force application. This constraint reflects the stability provided by surrounding maxillary anatomy during actual clinical loading.

Occlusal Load (B): A vertical occlusal force of 200 N was applied in the direction shown by the red arrow. This load replicates average physiological masticatory forces exerted on anterior teeth during function. The force vector is aligned perpendicular to the occlusal plane and directed axially toward the long axis of the implant-abutment assembly to ensure uniform loading conditions.

This setup ensures consistent and reproducible simulation conditions across all FEM models analysed in the study. Maintaining the same direction and magnitude of force across different implant-angulation-abutment combinations allowed for accurate comparison of stress distribution patterns in the cortical bone, cancellous bone, implant, and abutment components.

The stress generated in and around the implant were then determined by performing the stress analysis using ANSYS R17 software, and the stress values obtained after analysis were evaluated in terms of Von-Mises equivalent stress levels. The mathematical solutions obtained in the results were converted into visual results which were characterized by colors ranging from blue to red, in which red represented the highest stress values and shades of blue represented the minimum stress values.

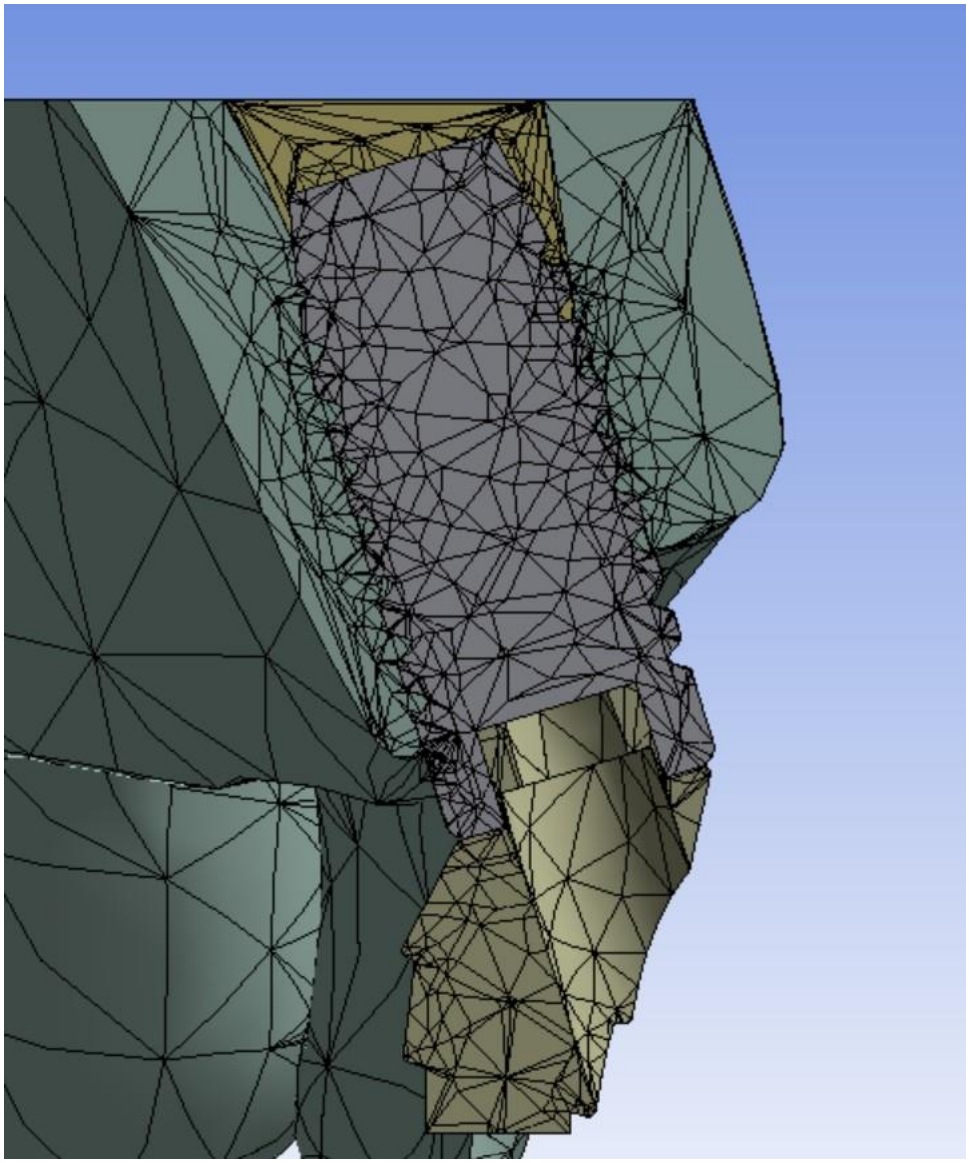


Figure 2 - Shows a cross-section mesh image of complete implant & transmucosal abutment

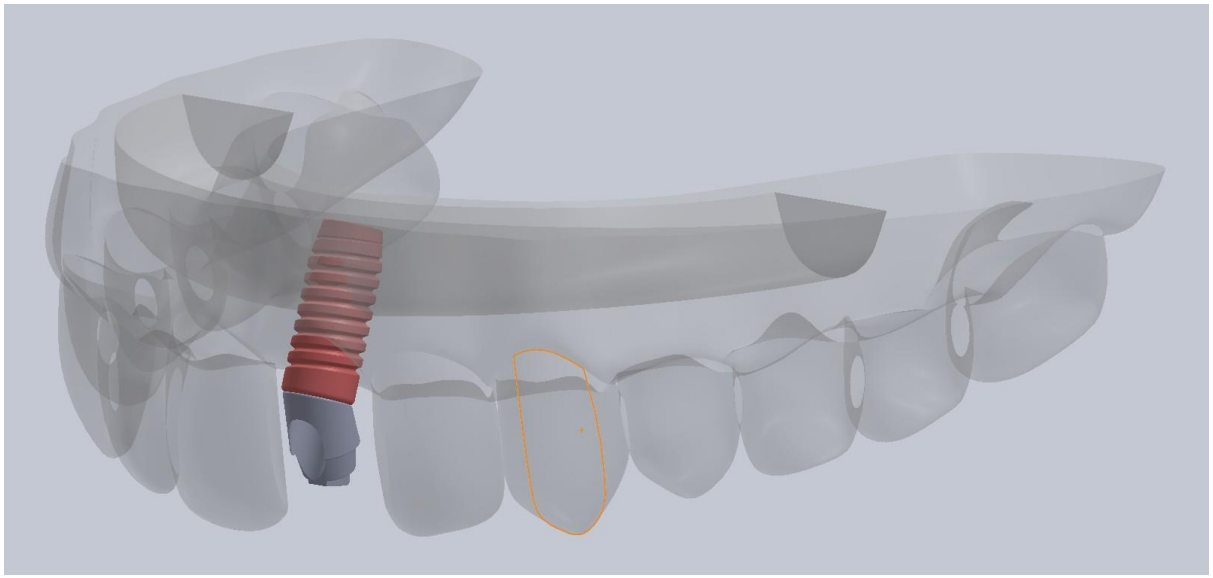


Figure 3(a) Represents the stimulated 3D models of maxilla with implant placed at 17° with 17°TMA

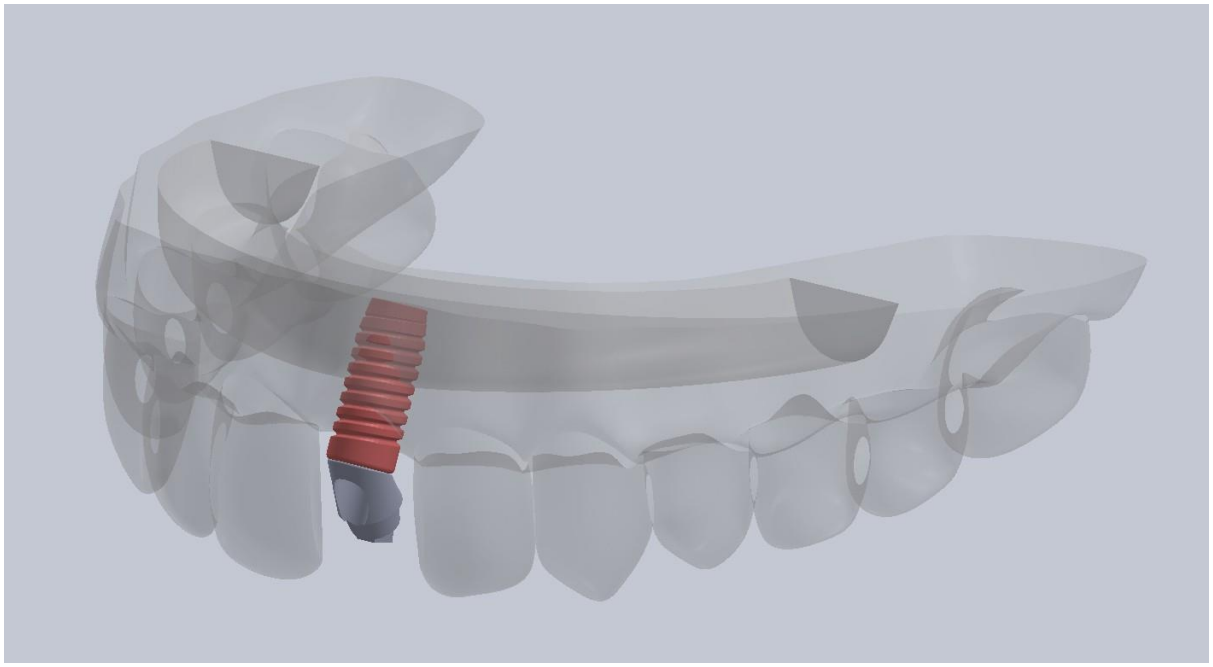


Figure 3(b) Represents the stimulated 3D models of maxilla implant placed at 17° with 30°TMA.

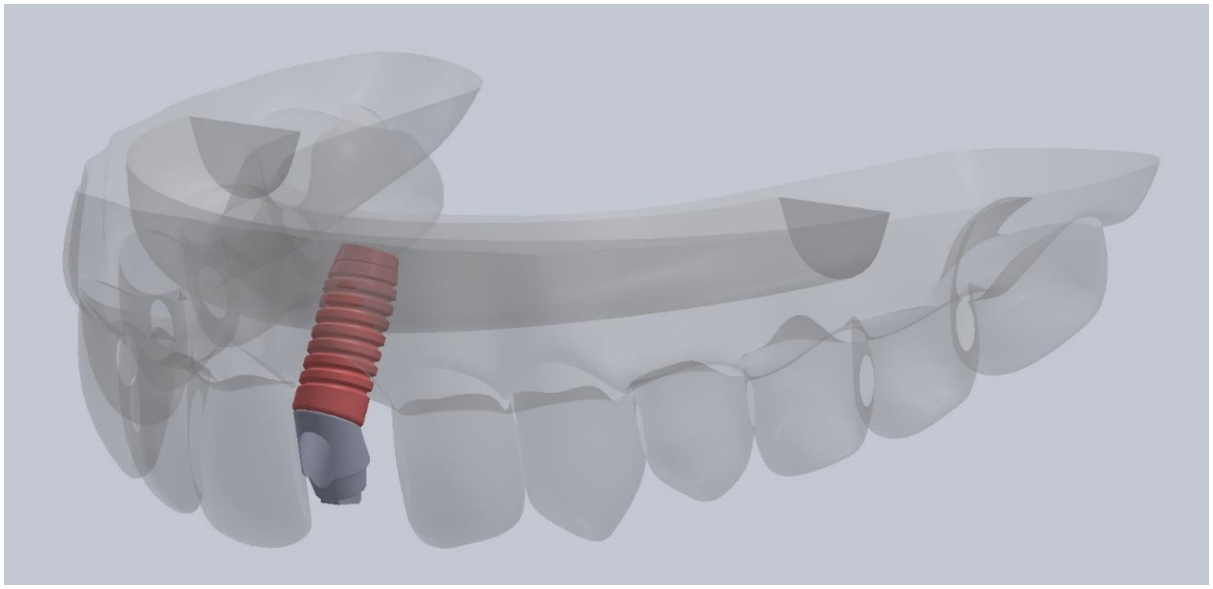


Figure 3(c) Represents the stimulated 3D models of maxilla with implant placed at 25° with 17°TMA

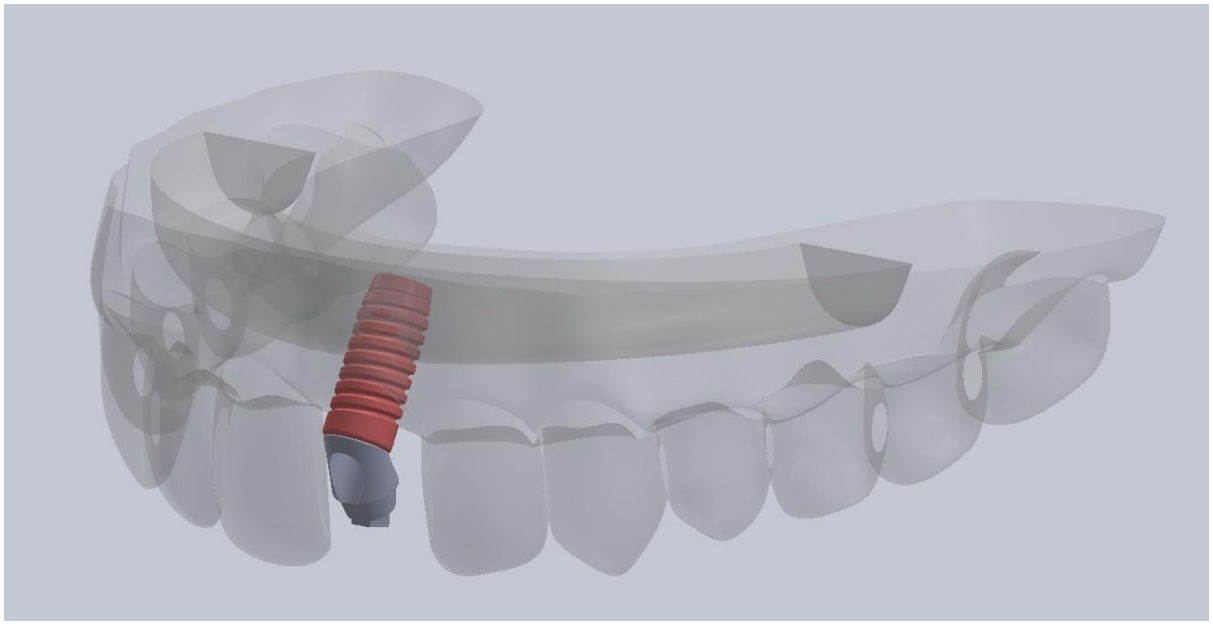


Figure 3(d) Represents the stimulated 3D models of maxilla with implant placed at 25° with 30°TMA

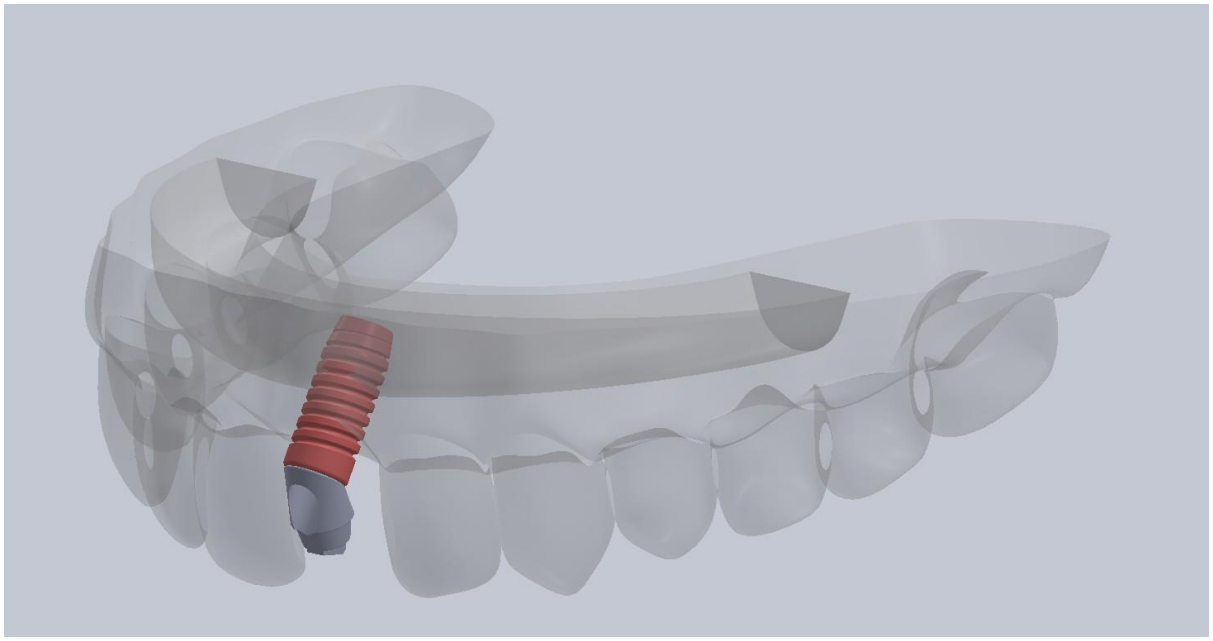


Figure 3(e) Represents the stimulated 3D models of maxilla with implant placed at 30° with 17°TMA.

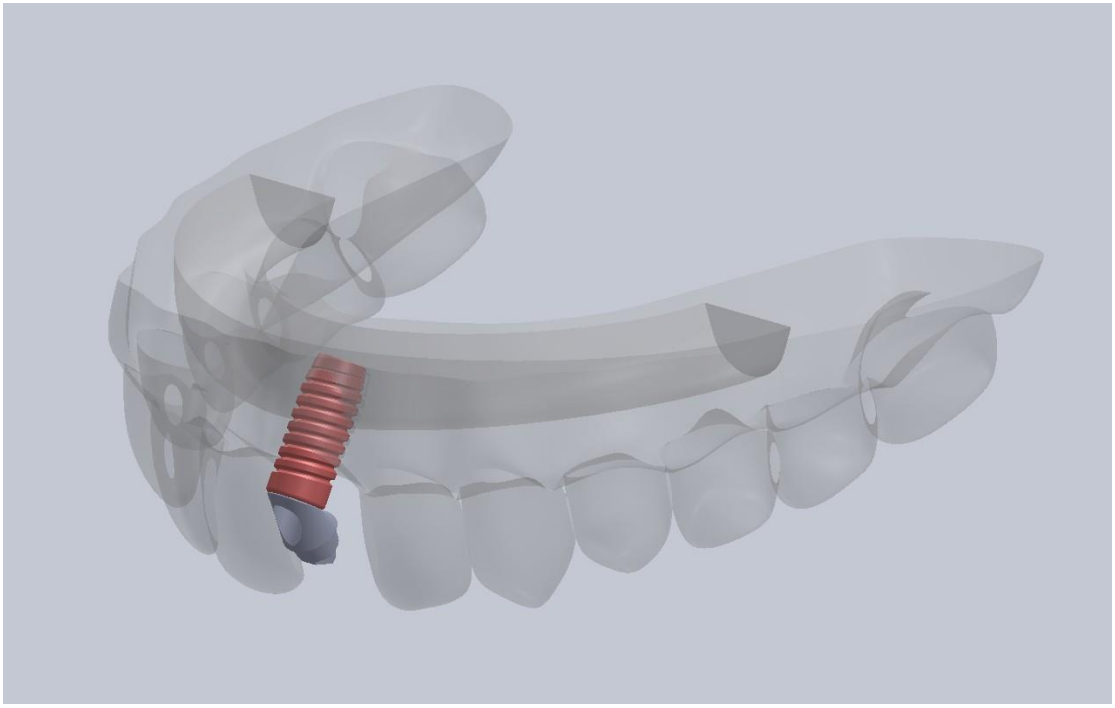


Figure 3(f) Represents the stimulated 3D models of maxilla with implant placed at 30° with 30°TMA.

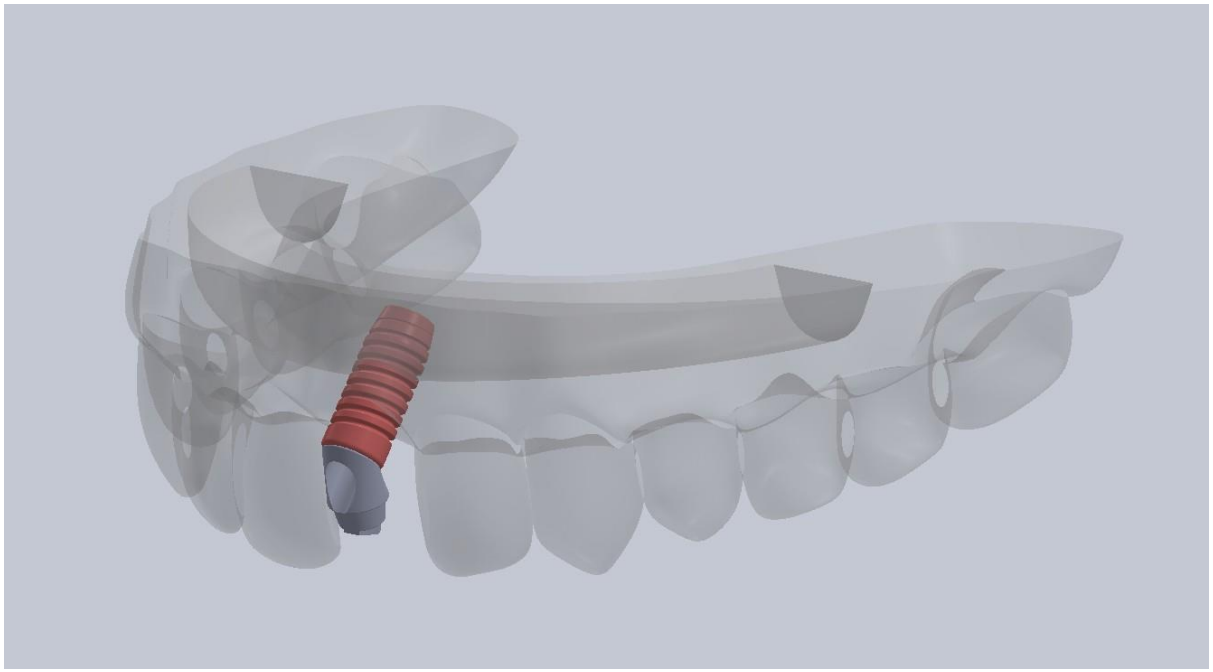


Figure 3(g) Represents the stimulated 3D models of maxilla with implant placed at 35° with 17°TMA.

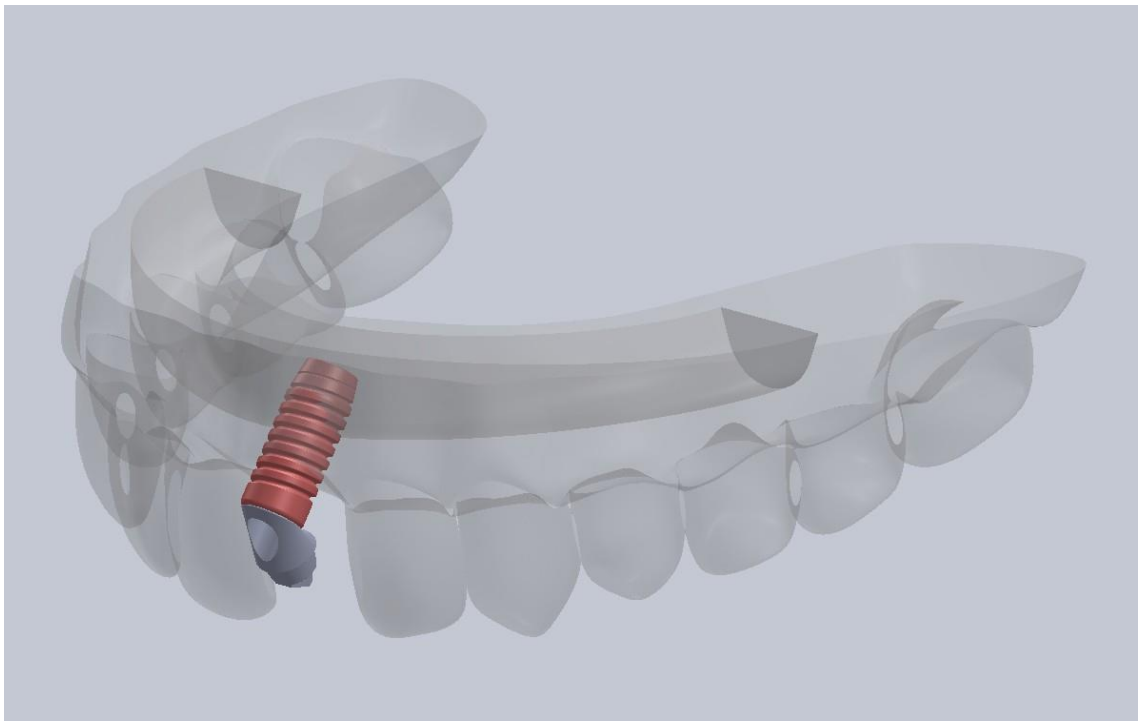


Figure3(h) Represents the stimulated 3D models of maxilla with implant placed at 35° with 30°TMA

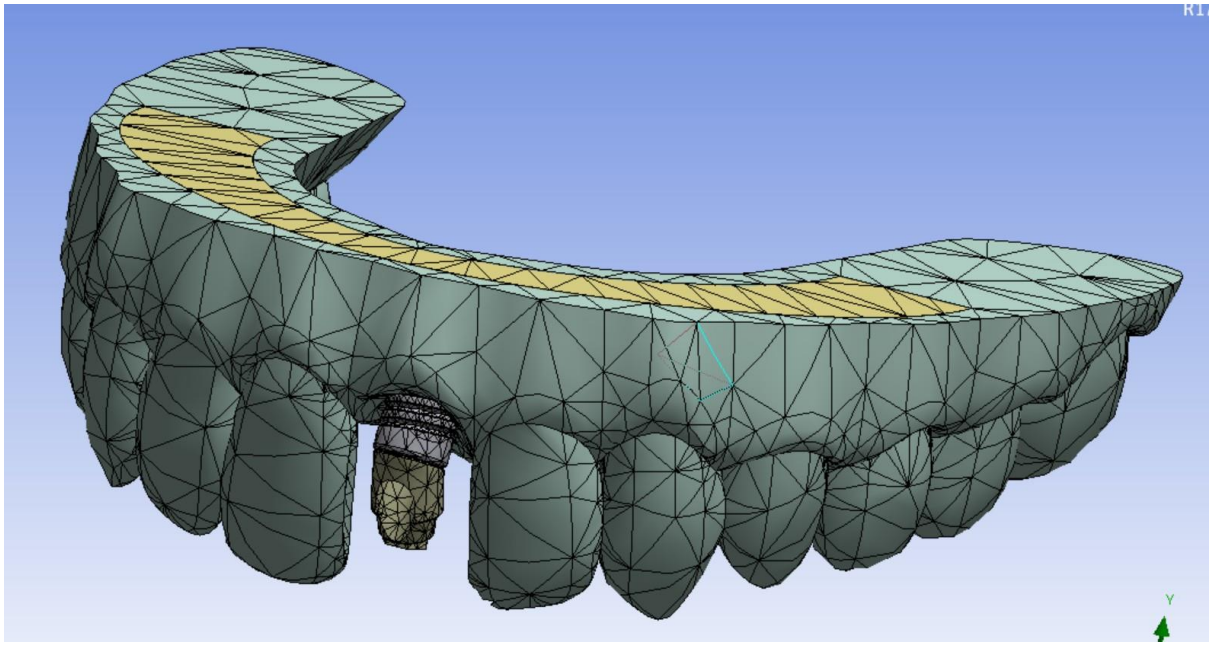


Figure 4(a) Represents the stimulated 3D models of maxilla with implant placed at 17° with 17°TMA.

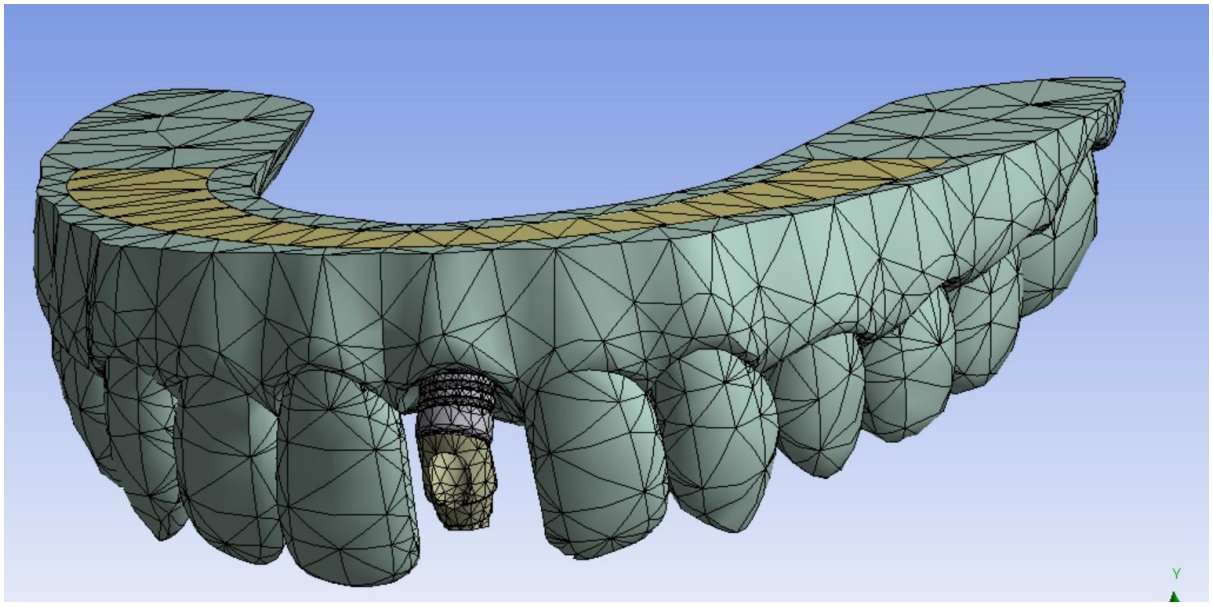


Figure4(b) Represents the stimulated 3D models of maxilla implant placed at 17° with 30°TMA.

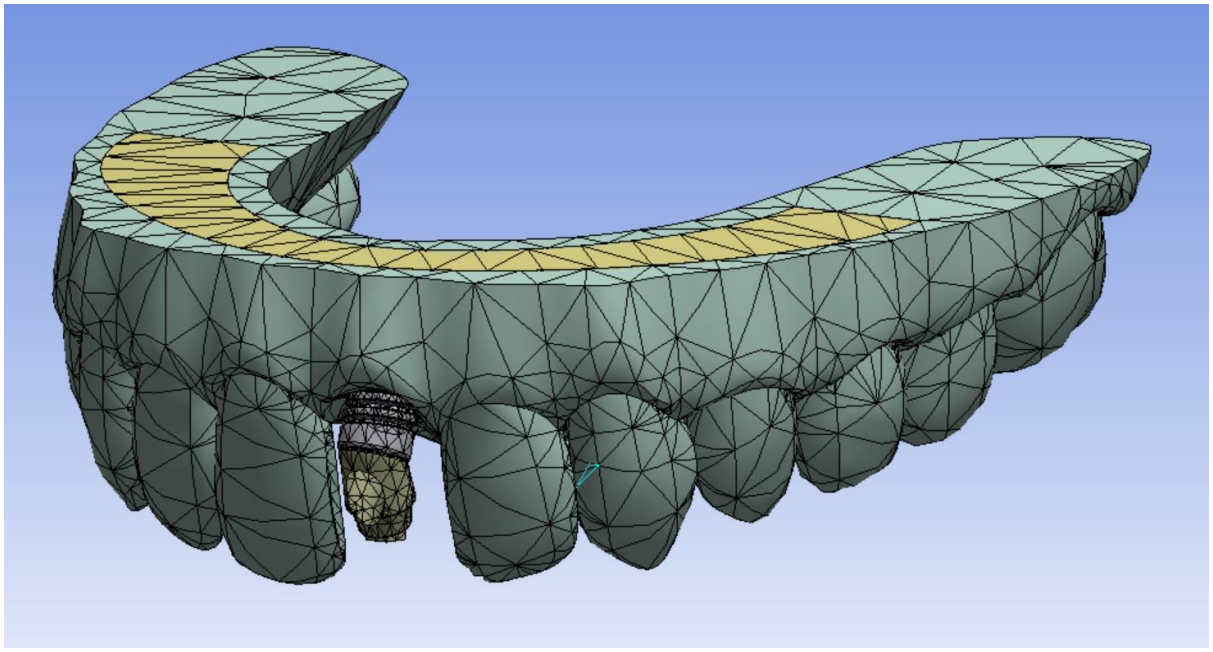


Figure4(c) Represents the stimulated 3D models of maxilla with implant placed at 25° with 17°TMA

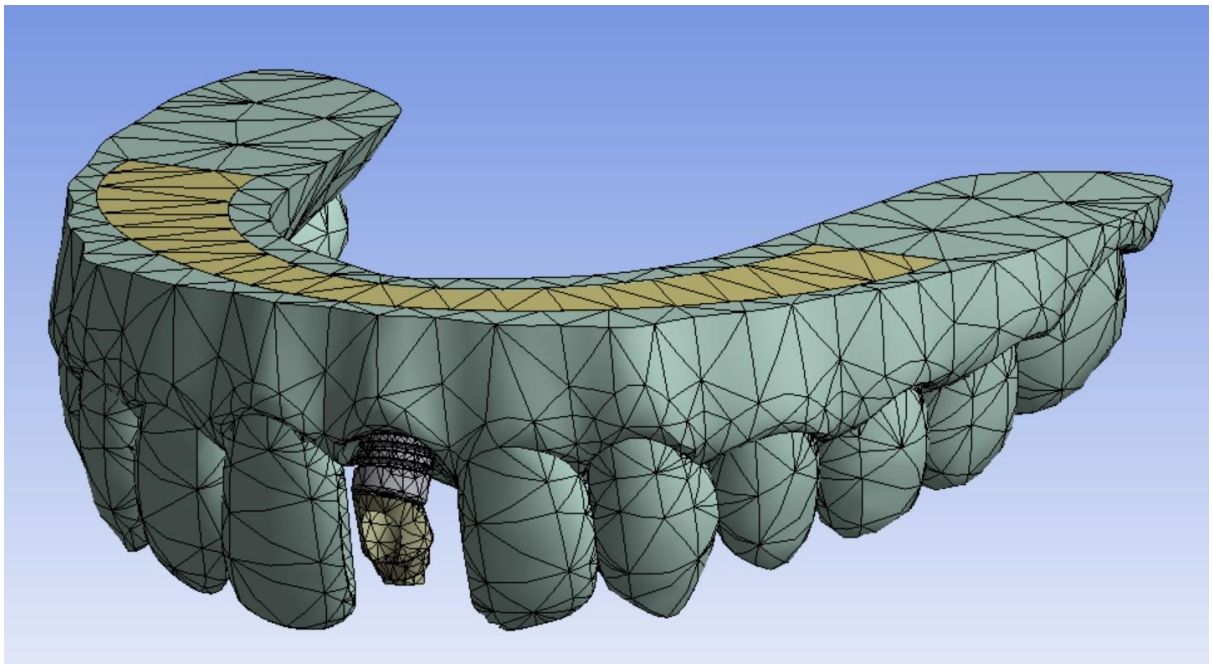


Figure 4(d) Represents the stimulated 3D models of maxilla with implant placed at 25° with 30°TMA.

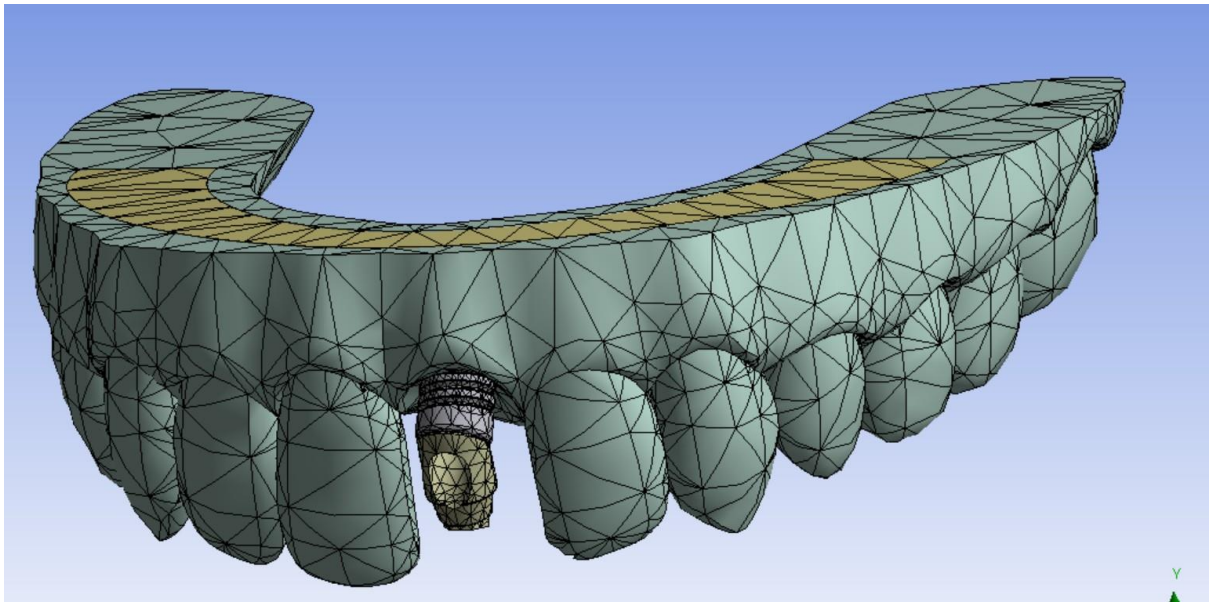


Figure 4(e) Represents the stimulated 3D models of maxilla with implant placed at 30° with 17°TMA.

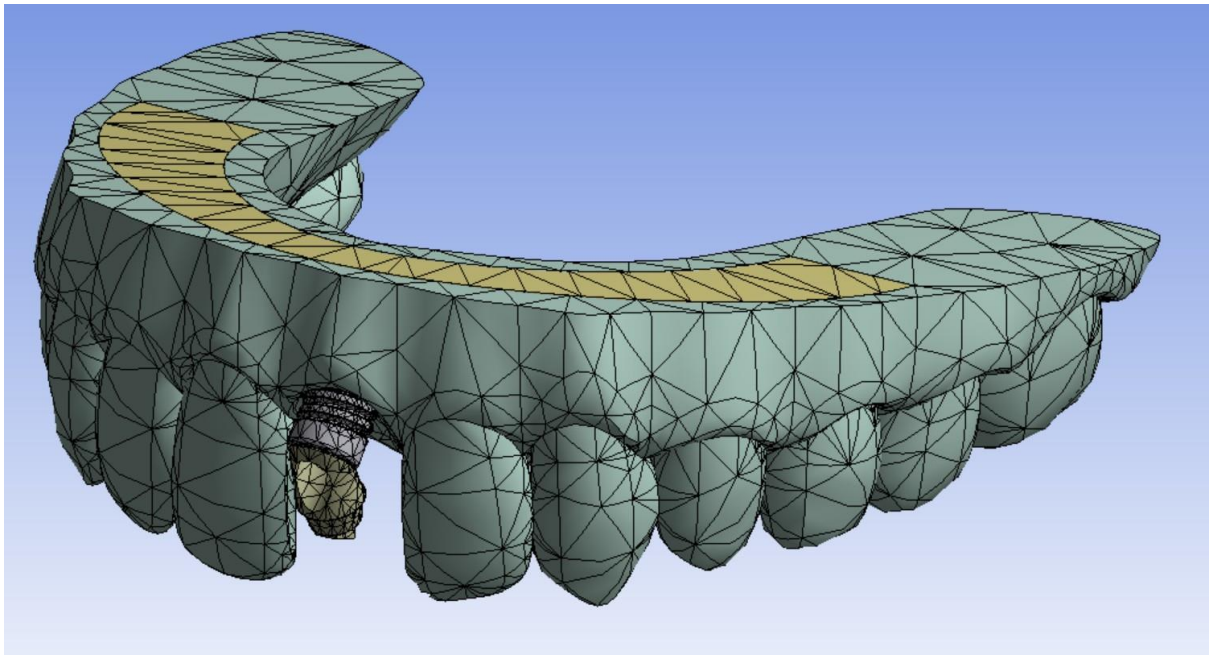


Figure 4(f) Represents the stimulated 3D models of maxilla with implant placed at 30° with 30°TMA.

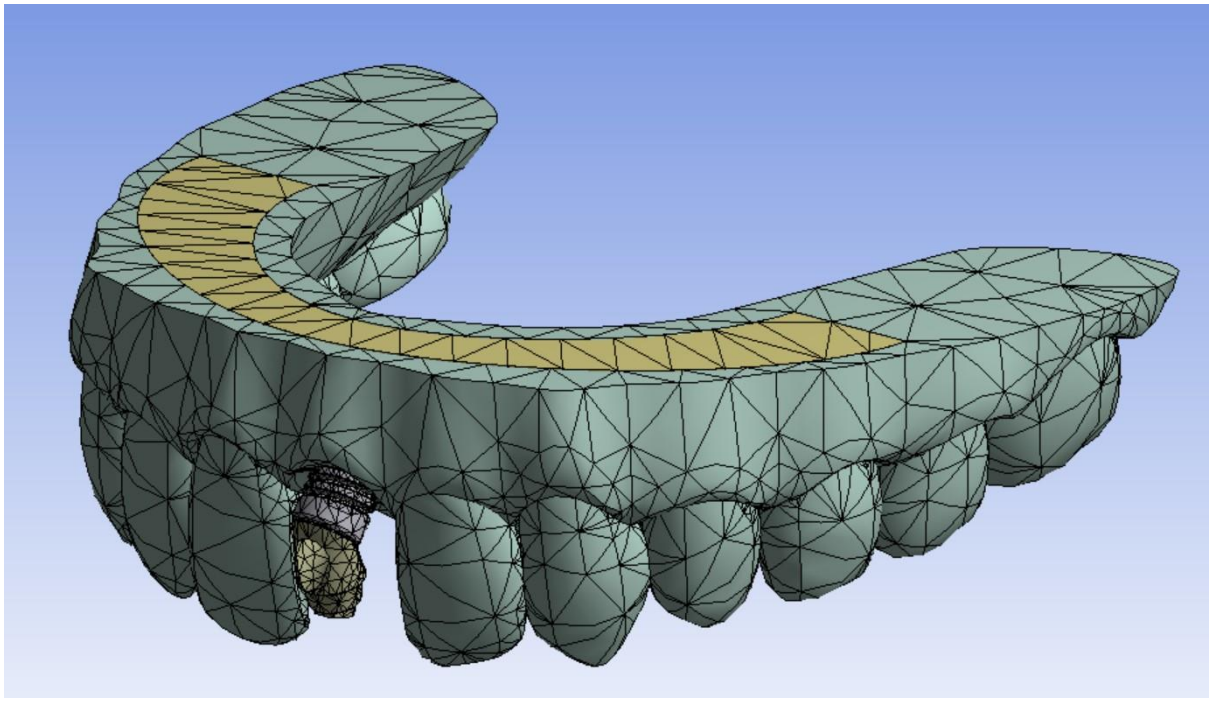


Figure4(g) Represents the stimulated 3D models of maxilla with implant placed at 35° with 17°TMA.

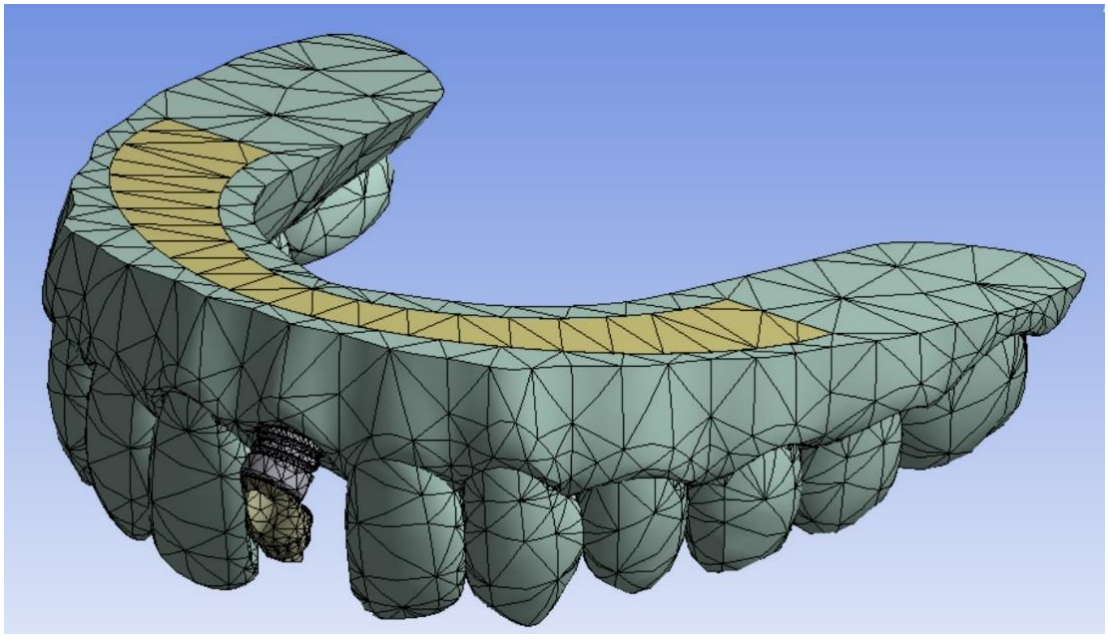


Figure 4(h) Represents the stimulated 3D models of maxilla with implant placed at 35° with 30°TMA.

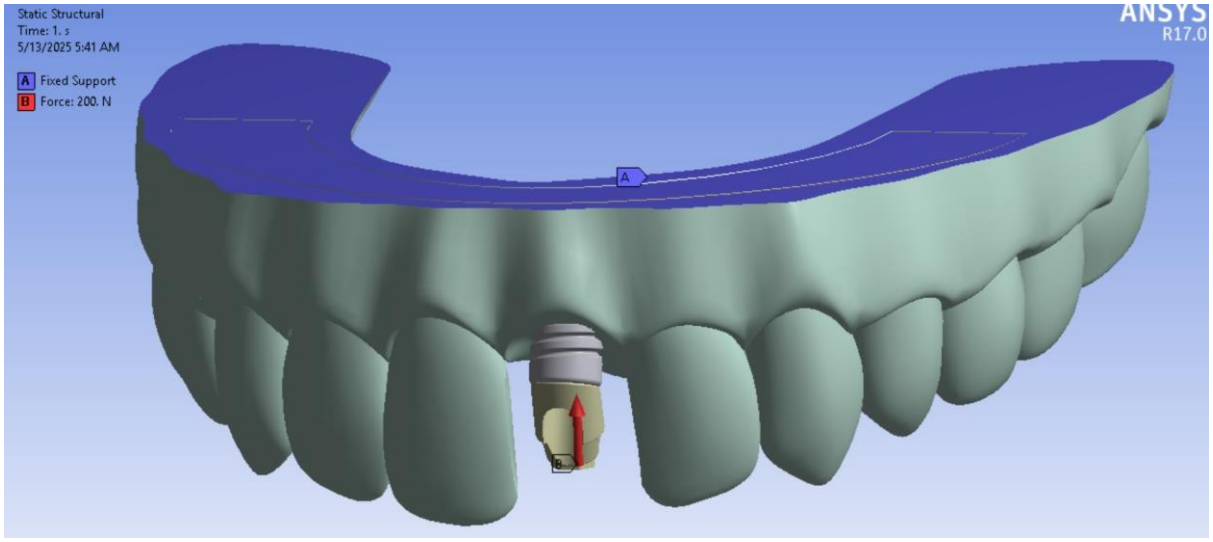


Figure5- Represents the dynamic loading of maxilla

Results and Observation

RESULTS AND OBSERVATIONS

The finite element analysis performed using ANSYS R17 software yielded detailed stress distribution patterns (Von Mises stresses) in and around the dental implants placed at various angulations and restored with 17° and 30° transmucosal abutments (TMA).

Model 1: Implant Tilt at 17° with Transmucosal Abutment Angulation at 17°

In Model 1, the stress distribution patterns revealed an efficient biomechanical environment resulting from the ideal alignment of both the implant and the transmucosal abutment at 17 degrees. This angulation match ensured a more axial and uniform force transmission throughout the implant-supported structure.

The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 3 and illustrated in Figures 6a, 6b, 6c, and 6d

Table3- Von Mises stress results for Model No. 1

MODEL 1	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**26.636**	**0.000**
CANCELLOUS BONE	**4.2652**	**0.000**
ABUTMENT	**216.92**	**0.411**
IMPLANT	**123.17**	**0.902**

The **cortical bone** exhibited a **maximum von Mises stress of 26.636 MPa**, with the stress concentrated primarily around the **crestal bone region adjacent to the implant neck**. This

concentration is **physiologically expected** due to the direct transmission of occlusal forces through the prosthetic components. However, the intensity of this stress remained **well within the tolerable physiological limits of cortical bone**, indicating that the crestal zone was not excessively overloaded. The surrounding cortical structure showed a **gradual and smooth dissipation of stress**, without the presence of sharp peaks or concentrated stress zones, reflecting a **biomechanically stable and favorable configuration**.

The **cancellous bone** demonstrated a more diffused stress pattern, with a **peak stress value of 4.2652 MPa** and a **minimum nearing zero (0.00031641 MPa)**. This stress was predominantly located in the **cervical region of the trabecular bone**, corresponding to the immediate interface below the implant crest. Despite being the more compliant and less dense component of the peri-implant bone, the cancellous region in this model did not exhibit any signs of excessive load or structural compromise. The stress distribution in this area indicated that the **axial alignment of the abutment with the implant** allowed the forces to be absorbed and dissipated efficiently through the bone architecture, **preserving its integrity** and reducing the risk of **microstructural remodelling or failure**.

Stress analysis of the transmucosal abutment showed a **maximum stress concentration of 216.92 MPa**, predominantly located near the **junction of the abutment with the implant fixture**. Despite being a zone of critical mechanical engagement, the stress levels remained **considerably lower than in mismatched models**. The **continuity in angulation** helped reduce the occurrence of **shear forces and torque**, which are common causes of prosthetic complications such as **screw loosening or component fracture**. The **smooth transfer of stress** along the length of the abutment without abrupt spikes reinforces the **mechanical advantage of using matching angulated components**.

The **implant body** itself experienced a **maximum stress of 123.17 MPa**, with stress regions localized at both the **apical and crestal aspects**. These areas, being primarily responsible for **anchorage and initial load reception**, naturally bear the brunt of occlusal forces. However, due to the **harmonious implant-abutment interface**, there was no evidence of **excessive bending moments or off-axis load accumulation**. The implant displayed a **uniform stress pattern** throughout its structure, with the **minimum stress recorded at 0.90297 MPa**, suggesting that the body of the implant functioned effectively under load **without risking structural fatigue or peri-implant bone strain**.

The **overall system performance in Model 1**, when evaluated holistically, reflected a **maximum stress value of 216.92 MPa across all components** and a **minimum stress of 0.00014176 MPa**. The **collective visual output**, indicated by a colour spectrum ranging from blue to red in the finite element simulation, demonstrated that **most of the model remained in the cooler stress zones**, with **isolated and moderate intensities** at the critical mechanical junctions. This clearly underlines the **biomechanical benefit of matching abutment angulation to implant tilt**, as seen in this configuration, which ensures **efficient stress transfer**, **preserves bone quality**, and **minimizes mechanical complications**.

In summary, **Model 1 served as the most biomechanically favourable condition** among the analysed models. The **uniform and controlled stress distribution** in cortical and cancellous bone, the **reduced mechanical strain on the abutment**, and the **stable load-bearing behaviour of the implant** all point to the **critical role of angulation matching in implant prosthodontics**. This model reinforces the recommendation that, wherever clinically feasible, the **angulation of the abutment should align with the implant tilt** to achieve **optimal long-term success and structural harmony**.

Model 2: Implant Tilt at 17° with Transmucosal Abutment Angulation at 30°

In this finite element analysis model (Model 2), the implant was angulated at **17 degrees** and restored with a **30-degree transmucosal abutment (TMA)**. The primary aim was to evaluate the biomechanical implications of a **mismatch between implant tilt and abutment angulation**, with particular emphasis on stress patterns generated in peri-implant bone, implant components, and the prosthetic interface.

The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 4 and illustrated in Figures 7a, 7b, 7c, and 7d

Table4- Von Mises stress results for Model No. 2

MODEL 2	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**29.484**	**8.250**
CANCELLOUS BONE	**3.4602**	**8.9217**
ABUTMENT	**544.84**	**0.70236**
IMPLANT	**235.01**	**1.1728**

The stress distribution within the **cortical bone** revealed a **maximum von Mises stress of 29.484 MPa** and a **minimum of 8.250 MPa**. The highest concentration of stress was observed at the **crestal bone region**, specifically on the labial aspect near the implant-abutment junction. This region is critical as it is the initial load-transferring interface. The elevated stress values in this area suggest that the angular discrepancy between the implant and abutment results in **non-axial loading**, which leads to **stress accumulation at the crestal level**. The

biomechanical implication of such stress concentration includes **marginal bone loss**, a known precursor to long-term implant failure. This observation is consistent with previous studies which confirm that the **greater the mismatch between implant and abutment angulation**, the higher the tendency for off-axis loading and crestal bone resorption.

The **cancellous bone** experienced **a maximum stress of 3.4602 MPa** and a **minimum of 8.9217e-5 MPa**. As expected, the overall stress magnitude in cancellous bone was significantly lower than in cortical bone, owing to its trabecular structure and lower modulus of elasticity. The stress appeared more **diffuse**, spreading around the inner surface of the implant cavity. However, even though cancellous bone can better absorb distributed loads, this model showed stress funnelling toward the cervical cancellous area—indicative of oblique force transfer caused by the angular misalignment. Although the values remain within physiological limits, **prolonged exposure to non-axial loading could jeopardize trabecular integrity**, leading to micromovements or progressive bone remodelling.

The **transmucosal abutment** showed the **highest stress concentration of all components** in this model, with a **maximum value of 544.84 MPa** and **minimum of 0.70236 MPa**. The location of peak stress was primarily in the **mid-section of the abutment**, near the transition zone to the implant platform. The sharp angular deviation between the abutment (30°) and implant (17°) likely introduced **significant bending forces and torque** at this critical interface. Such excessive stress, especially under cyclic or parafunctional loading, may lead to **mechanical complications** like abutment loosening, screw fracture, or deformation. These findings support existing literature that cautions against using highly angled abutments on less-tilted implants without careful load analysis.

The **implant** in this configuration registered a **maximum stress of 235.01 MPa** and a **minimum of 1.1728 MPa**. Stress was primarily localized at the **apical and crestal regions**,

with the crestal side affected due to the load transferred via the steeply angled abutment. The **oblique nature of loading**, resulting from the angular misalignment, exacerbated stress concentration at the implant-bone interface and within the implant body. If such forces persist over time without compensation by proper prosthetic design, the implant may face **material fatigue, microfractures**, or **loss of osseointegration**.

Overall Stress Distribution (All Structures)

When evaluating the model as a whole, the **maximum stress across the entire system was 544.84 MPa**, with a **minimum of 8.250e-5 MPa**. The **abrupt transition of force** from the abutment to the implant and surrounding bone created **non-uniform stress distribution**, with distinct hotspots of high stress and areas of low load sharing. This uneven pattern indicates **biomechanical inefficiency**, where certain structures are overloaded while others are underutilized in load transmission. In clinical terms, such patterns predispose the system to early fatigue, micro-movement, or even component failure if left unaddressed.

Model 2 demonstrates the **risks associated with mismatching the angulation of abutments to the implant tilt**. Though a 30° TMA may be chosen for prosthetic convenience or esthetic correction, its use on a 17° tilted implant introduces considerable biomechanical challenges. The elevated stress in the abutment and cortical bone emphasizes the **need for abutment angulation to match or closely approximate the implant's tilt** to ensure optimal load transfer and long-term peri-implant health. The findings reinforce the **importance of prosthetic planning during implant placement**, and suggest that clinical protocols should favor matching abutment angles to implant tilt whenever feasible, especially in anterior regions where occlusal forces and esthetics are both critical.

Model 3: Implant Tilt at 25° with Transmucosal Abutment Angulation at 17°

In Model 3, where the implant was angulated at 25 degrees and restored with a 17-degree transmucosal abutment, the finite element analysis revealed a distinct pattern of biomechanical response indicative of angular mismatch. The von Mises stress in the cortical bone reached a **maximum of 24.007 MPa**, while the **minimum stress was recorded as low as 0.00023128 MPa.** The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 5 and illustrated in Figures 8a, 8b, 8c, and 8d**.**

Table5- Von Mises stress results for Model No. 3

MODEL 3	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**24.007**	**0.000**
CANCELLOUS BONE	**5.5897**	**0.000**
ABUTMENT	**234.52**	**0.170**
IMPLANT	**131.98**	**0.784**

The **highest stress concentration** was observed at the **crestal region of the cortical bone** surrounding the **implant neck**, a site that commonly endures **peak loads during occlusal function**. Although the stress value remained within the **physiologic limit**, the **mismatch between the implant and abutment angulation** resulted in **slightly elevated stress zones** when compared to **Model 1**. The moderate peak suggests that while the bone is not overloaded, the deviation in angulation introduced additional strain, particularly at the labial aspect of the crestal bone.

The **cancellous bone** displayed a more diffuse stress distribution pattern with a **maximum von Mises stress of 5.5897 MPa** and a **minimum of 0.00066589 MPa**. These stresses were largely concentrated around the **cervical portion of the implant**, indicating that the internal trabecular architecture of the bone was being subjected to **non-axial loading** as a result of the **angular offset** between the abutment and implant. The **increased magnitude** compared to both **Model 1 and 2** suggests that even moderate mismatches can lead to greater force transmission into the cancellous core, potentially resulting in bone remodeling or microstructural fatigue under repeated loading.

The **abutment structure**, subjected to forces transferred through a **misaligned axis**, demonstrated a **maximum stress concentration of 234.52 MPa** and a **minimum of 0.17032 MPa**. The stress was localized primarily around the **connection area between the abutment and implant platform**, indicating **torque and shear forces** acting at the interface. The **elevated value** reflects the biomechanical consequences of **restoring a more steeply tilted implant with a less angulated abutment**. This mismatch likely induced **off-axis forces** that translated into **bending moments** within the abutment, increasing the likelihood of **mechanical complications** such as **screw loosening** or **material fatigue**.

Within the **implant itself**, the **maximum von Mises stress was 131.98 MPa** and the **minimum was 0.78479 MPa**. The stress was more pronounced at the **apical and crestal regions**, correlating with the **altered transmission pathway** of occlusal forces. Unlike in the **matched configuration (Model 1)**, the implant in this case experienced a **greater degree of non-uniform stress distribution**, primarily attributed to the **misalignment in angulation**, which compromised **axial load-bearing capacity** and introduced **oblique forces**. While the implant remained **within material tolerance**, the **increased stress** hints at potential for **micro-movement or fatigue** over time.

When analysing the system holistically, the entire model recorded a **maximum stress of 234.52 MPa** and a **minimum of 0.00023128 MPa**. The **global stress profile** reflected a **non-uniform distribution** with evident **stress intensification at the implant-abutment junction** and the **adjacent bone structures**. The **red zones** visible in the finite element visualization underscore the **mechanical inefficiency** caused by the **angulation mismatch**, with stress values **higher than those in Model 1**. Although not as extreme as in **highly divergent angulation combinations**, the biomechanical behavior of Model 3 illustrates the **subtle yet significant impact of even an 8-degree mismatch**.

In summary, **Model 3 demonstrated** that restoring a **25-degree tilted implant with a 17-degree abutment** results in **measurable stress intensification** across both hard and prosthetic components. The **cortical and cancellous bone** remained within **acceptable physiological thresholds** but showed **early signs of strain** due to **non-axial load transfer**. The **abutment and implant**, particularly at their junction, **bore the brunt of mechanical imbalance**, confirming the **clinical importance of minimizing angular discrepancy** to promote **favourable load distribution** and ensure **long-term prosthetic stability**.

Model 4: Implant Tilt at 25° with Transmucosal Abutment Angulation at 30°

Model 4 represents a **moderate angular mismatch scenario**, where a **25-degree tilted implant** was restored using a **30-degree transmucosal abutment**. This 5-degree difference introduced mild off-axis loading conditions. The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 6 and illustrated in Figures 9a, 9b, 9c, and 9d.

Table6- Von Mises stress results for Model No. 4

MODEL 4	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**22.47**	**0.000**
CANCELLOUS BONE	**7.128**	**0.000**
ABUTMENT	**506.73**	**0.319**
IMPLANT	**205.73**	**1.054**

The **cortical bone** exhibited a **maximum stress of 22.47 MPa**, with primary stress concentration at the **crestal bone region near the implant shoulder**, particularly on the **labial aspect**. While this level of stress is within the **physiological tolerance** of cortical bone, it reflects a **mild intensification of force due to the angular mismatch**. The stress was less severe than in more mismatched configurations but notably higher than in perfectly aligned models like Model 1, suggesting a **non-axial transmission of load** despite partial alignment.

In the **cancellous bone**, the **maximum von Mises stress reached 7.128 MPa**. This was focused primarily in the **cervical cancellous region**, around the implant neck. Due to its porous and elastic nature, cancellous bone is more vulnerable to microdamage under repeated stress. Although the values remain within biomechanical safety limits, the presence of **oblique force transmission** increases the likelihood of **bone remodelling** or **early trabecular strain**, especially under long-term cyclic loading.

The **abutment** endured the **highest stress concentration in the entire system**, with a **peak value of 506.73 MPa**. This extremely high value suggests that the **implant-abutment interface**, particularly the **abutment collar**, was under significant **bending and shear forces**. The 5° mismatch between the implant and abutment likely introduced **residual torque**, making

this region susceptible to **mechanical complications**, such as **screw loosening, thread stripping, or fatigue failure**. Among all the models tested, this configuration places the **prosthetic connection under one of the most severe stress conditions**.

Within the **implant body**, the **maximum stress was recorded at 205.73 MPa**, with stress localized mainly at the **apical and crestal thread regions**. This stress level, while lower than in the abutment, was **elevated compared to better-aligned models**, pointing to **inefficient axial load transmission**. The presence of **non-uniform stress zones** raises concerns for **implant fatigue, micromotion**, or potential **loss of osseointegration** in long-term clinical scenarios.

Evaluating the **overall system**, the simulation displayed a **maximum stress of 506.73 MPa**, with red zones dominating the **abutment-implant interface** on the finite element stress map. The rest of the model showed cooler (blue-green) zones, indicating that **biomechanical strain was highly localized** due to the angular misalignment.

Model 4 clearly demonstrates that even a **minor angular mismatch of 5°**, such as restoring a **25° tilted implant** with a **30° abutment**, can lead to **significant biomechanical consequences**, particularly on the **prosthetic interface**. Although **cortical and cancellous bone** stress remained within **safe limits**, the **abutment experienced extreme stress**, indicating a potential point of mechanical failure. These findings highlight the **importance of achieving close angular compatibility** between implant and abutment components. Even small discrepancies, especially in **anterior or high-load regions**, must be managed with careful prosthetic planning to ensure **long-term stability and success** of implant-supported restorations.

MODEL 5-Implant placed at 30° with 17° Transmucosal Abutment (TMA)

Model 5 features a **significant angular mismatch**, where a **30-degree tilted implant** is restored using a **17-degree transmucosal abutment (TMA)**, resulting in a **13-degree discrepancy**. The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 7 and illustrated in Figures 10a, 10b, 10c, and 10d.

Table7- Von Mises stress results for Model No. 5

MODEL 5	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**30.433**	**0.000**
CANCELLOUS BONE	**6.400**	**0.000**
ABUTMENT	**467.8**	**0.299**
IMPLANT	**139.66**	**0.296**

The **cortical bone** displayed a **maximum stress of 30.433 MPa**, indicating one of the highest levels observed in this study. Stress was intensely concentrated at the **labial crestal region** around the **implant neck**, where the abrupt angulation mismatch generated **non-axial load transmission**. The vivid red and yellow zones in the simulation maps pointed to a potential risk of **crestal bone overload**, which, if exposed to repetitive occlusal forces, could lead to **early marginal bone resorption** despite the stress remaining within the bone's ultimate tolerance.

The **cancellous bone** experienced a **maximum stress of 6.4007 MPa**, primarily near the **cervical trabecular interface**. The stress was moderately high and more diffusely spread through the cancellous core than in aligned models. The **improper redirection of occlusal**

forces due to the angular mismatch forced the cancellous bone to bear load unevenly, increasing the likelihood of **bone fatigue, microfractures, or remodeling** over time under clinical conditions.

The **abutment** was revealed to be the most critically stressed component, with a **maximum stress concentration of 467.8 MPa** and a **minimum of 0.2997 MPa**. The **implant-abutment interface** absorbed a majority of the force, especially at the **abutment collar**, where **torque and bending moments** were most pronounced. The **13° mismatch** significantly compromised the biomechanical harmony, creating a scenario with a **high risk of mechanical failure**, including **screw loosening, structural fatigue**, or **fracture of the abutment** under cyclic loading.

The **implant body** exhibited a **maximum stress of 139.66 MPa**, with notable stress localized at the **apical and crestal threads**. The axial misalignment caused load to be transmitted **obliquely**, placing strain on areas of the implant not optimized for peak stress bearing. This could potentially result in **implant fatigue, deformation**, or **progressive micromovement**, especially in patients with high masticatory loads or parafunctional habits.

Assessing the **entire model**, the **peak overall stress** reached **467.8 MPa**, and the **minimum was 0.00030781 MPa**. The most vulnerable zone remained the **abutment-implant junction**, followed by the adjacent cortical bone and implant body. The **global stress visualization** displayed a distinctly **asymmetrical and uneven distribution**, highlighting a clear **mechanical imbalance** due to the steep angulation discrepancy.

Model 5 confirms the **biomechanical drawbacks of restoring a 30° tilted implant with a 17° abutment**. The **13-degree mismatch** resulted in **significantly elevated stress**, especially at the **prosthetic interface and crestal bone**. Although the stress levels in bone remained within acceptable limits, the **sharp concentration and direction of force vectors** placed the

abutment under substantial mechanical threat, making it a **likely point of failure** if such a configuration is used clinically. These results strongly advocate for **minimizing angulation mismatches** in implant restorations to ensure **balanced load transfer, protect peri-implant bone health**, and promote **long-term functional success** of the prosthesis.

MODEL 6-Implant placed at 30° with 30° Transmucosal Abutment (TMA)

Model 6 represents a biomechanically optimized scenario in which the implant and abutment share the **same angulation of 30°**, creating a **matched configuration** that promotes axial force distribution. Despite the high implant tilt, the results show that this angular harmony plays a pivotal role in influencing the internal stress patterns across the peri-implant region and prosthetic components.

The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 8 and illustrated in Figures 11a, 11b, 11c, and 11d.

Table8- Von Mises stress results for Model No. 6

MODEL 6	***MAXIMUM VALUE***	***MINIMUM VALUE***
CORTICAL BONE	**22.098**	**0.000**
CANCELLOUS BONE	**5.378**	**0.000**
ABUTMENT	**773.16**	**1.474**
IMPLANT	**276.05**	**0.441**

In the **cortical bone**, the maximum von Mises stress was recorded at **22.098 MPa**, with a minimum value as low as **0.00020167 MPa**. The stress distribution was relatively **uniform**

and concentrated primarily at the crestal bone near the implant neck, but without the excessive hotspots seen in mismatched configurations. This indicates that, although 30° tilt introduces a higher off-axis component, the matched abutment effectively redirects much of the occlusal force axially, reducing unnecessary lateral stress. The peak value remained **well within the physiological tolerance of cortical bone**, minimizing the risk of crestal bone loss or resorption under clinical loading.

In the **cancellous bone**, the stress values were similarly favorable, with a **maximum stress of 5.3786 MPa** and **minimum at 0.00076457 MPa**. The inner trabecular region around the implant exhibited **diffuse but moderate stress levels**, which are expected with deep seating and adequate angular compensation. These findings suggest **effective load dampening**, with reduced risk of microstrain-induced bone remodeling when compared to unmatched angulation scenarios.

The **abutment**, however, experienced the **highest mechanical strain in the entire system**, with a maximum von Mises stress of **773.16 MPa** and a minimum of **1.4746 MPa**. This represents **the highest abutment stress among all evaluated models**, even exceeding Model 5, which had a 13° mismatch. The reason likely lies in the **high implant tilt**, which, although compensated, still results in a significant load magnitude transmitted through the abutment. The stress concentration was sharply localized at the **implant-abutment collar**, a known point of bending and torsional stress. These findings highlight a crucial mechanical consideration: **even with angular match, extreme tilt magnifies load intensity**, and abutment design and material selection become essential to avoid fatigue failure.

The **implant body** displayed a **maximum stress of 276.05 MPa**, with a **minimum of 0.44182 MPa**. The highest stress was again concentrated around the **apical and crestal thread regions**, consistent with the load transfer behavior under oblique forces. While the values are higher

than those observed in lower-angle models, the distribution was more centralized and symmetric, implying **controlled force dispersion due to the matched abutment configuration**.

From a system-wide perspective, Model 6 recorded the **highest total stress in the entire study**, with the **maximum von Mises stress peaking at 773.16 MPa** and a **minimum of 0.00020167 MPa**. The simulation showed a vivid red zone focused on the abutment, underscoring its role as the mechanical bottleneck in this model. While the bone components experienced reduced stress due to the angulation match, the **prosthetic interface absorbed the brunt of occlusal forces**, demanding attention to **screw design, tightening torque, and material resilience**.

In summary, Model 6 demonstrated that **matching abutment angulation to a highly tilted implant (30°)** results in **efficient force redirection**, thereby reducing stress in the surrounding bone structures. However, this comes at the cost of **extremely high stress on the abutment**, making it the **most critical mechanical point in the system**. These findings emphasize that while angular alignment is essential, **high-tilt configurations still pose mechanical risks**, and such cases should be reinforced with high-performance materials and robust design to prevent prosthetic failure over time.

Model 7 - : Implant Tilt at 35° with Transmucosal Abutment Angulation at 17°

Model 7 represents a **significant angular mismatch**, in which a **35-degree tilted implant** is restored using a **17-degree transmucosal abutment (TMA)**, resulting in an **18-degree difference**. This configuration reflects a steep biomechanical discrepancy.

The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 9 and illustrated in Figures 12a, 12b, 12c, and 12d.

Table9- Von Mises stress results for Model No. 7

MODEL 7	*MAXIMUM VALUE*	*MINIMUM VALUE*
CORTICAL BONE	**33.162**	**0.000**
CANCELLOUS BONE	**4.848**	**0.000**
ABUTMENT	**442.29**	**0.130**
IMPLANT	**225.42**	**0.332**

The **cortical bone** experienced a **maximum stress of 33.162 MPa**, among the highest across all models. The **stress was heavily concentrated at the crestal region**, especially on the **labial side of the implant neck**, reflecting the impact of the **off-axis load transmission** due to the pronounced mismatch. Although still within cortical bone's physiological endurance limits, this elevated stress level suggests a **potential risk of crestal bone overload**, especially under cyclic masticatory forces. Over time, this could compromise **marginal bone stability** or induce **bone resorption**.

In the **cancellous bone**, the **maximum stress recorded was 4.848 MPa**, notably lower than that of the cortical bone but still significant given the **lower density and elastic modulus** of trabecular bone. The stress was localized in the **cervical region**, and although within acceptable limits, sustained loading in this zone could potentially lead to **trabecular fatigue or bone remodeling**, especially in patients with reduced bone quality.

The **abutment** endured a **peak stress of 442.29 MPa**, making it one of the **most mechanically vulnerable components** in this configuration. The sharp angular mismatch introduced

substantial torque and bending moments at the **abutment collar and its junction with the implant platform**. These forces created intense stress zones, especially at the transition interface, posing risks such as **screw loosening, deformation**, or **fracture of the abutment structure**. Clinically, such stress concentration is alarming, especially in load-bearing zones like the anterior maxilla, where aesthetics and function are both critical.

The **implant body** registered a **maximum stress of 225.42 MPa**, with the load primarily affecting the **apical and mid-thread regions**. The **non-axial transmission of forces** due to angular misalignment shifted the stress away from optimal load-bearing surfaces, which may increase the potential for **implant fatigue**, **micro-movement**, or **structural compromise** over extended periods of use.

When evaluating the **overall system**, the **maximum stress remained 442.29 MPa**, again highlighting the **abutment as the most critical biomechanical hotspot** in this model. The red zone visualized at the **implant-abutment interface** confirmed a **high concentration of biomechanical stress**, while the rest of the model displayed relatively cooler zones, suggesting that the mismatch heavily burdened specific prosthetic components rather than the entire structure.

Model 7, involving a **35° implant restored with a 17° abutment**, clearly demonstrates the **adverse biomechanical effects of substantial angular mismatch**. The **18-degree discrepancy** resulted in **excessive stress** at the **implant-abutment junction**, alongside elevated loads in both cortical and cancellous bone. While bone stress levels stayed within physiological tolerance, the prosthetic interface — particularly the **abutment** — absorbed an unsustainable share of the mechanical load, making it prone to **mechanical failure**. This configuration underscores the **importance of minimizing angular discrepancies**, especially

in the anterior region, to promote **stress harmony**, **implant longevity**, and **prosthetic stability**.

Model 8: Implant Tilt at 35° with Transmucosal Abutment Angulation at 30°

In this finite element analysis model, the stress distribution was evaluated across the cortical bone, cancellous bone, abutment, and implant when the implant was placed at a **30° tilt** and restored with a **35° transmucosal abutment**. The finite element analysis conducted using ANSYS R17 under a 200 N vertical load yielded the Von Mises stress results, as presented in Table 10 and illustrated in Figures 13a, 13b, 13c, and 13d.

Table10 - Von Mises stress results for Model No. 8

MODEL 8	***MAXIMUM VALUE***	***MINIMUM VALUE***
CORTICAL BONE	**24.256**	**0.000**
CANCELLOUS BONE	**3.2516**	**0.000**
ABUTMENT	**1072.8**	**1.672**
IMPLANT	**273.05**	**0.377**

Within the **cortical bone**, the **maximum equivalent (von-Mises) stress** recorded was **24.256 MPa**, while the minimum stress was negligible. The stress was most concentrated at the **crestal region**, surrounding the implant neck, where the implant engaged the cortical layer. The magnitude remained within acceptable physiological limits, indicating that even at higher angulations, the cortical bone is capable of managing the stress with minimal risk of mechanical failure or bone resorption.

In the **cancellous bone**, the **maximum stress observed was 3.2516 MPa**. The stress in this region was significantly lower compared to the cortical bone, with the highest stress concentrated adjacent to the implant apex and around the inner threads. Given the more elastic and less dense nature of cancellous bone, the stress remains well below levels that would typically lead to trabecular collapse or bone fatigue.

The **abutment** showed the **highest stress concentration among all components**, reaching a **peak of 1072.8 MPa**. This extremely elevated stress indicates a **critical biomechanical overload**, particularly at the junction between the implant and the abutment shoulder. The stress concentration was sharply localized at the **cervical collar of the abutment**, which serves as a biomechanical fulcrum in high angulation mismatch scenarios. This suggests that a 35° abutment on a 30° implant may introduce excessive shear forces at this interface.

Similarly, the **implant** itself exhibited **maximum stress of 273.05 MPa**, with the majority of stress localized at the **apical threads and neck region**. The stress pattern illustrates how misaligned abutment-implant configurations can generate torque and off-axis loading, leading to elevated strain along the implant body, particularly in the apical and mid-thread zones.

When observing the **entire model (ALL components)**, the **maximum overall stress recorded was also 1072.8 MPa**, confirming that the abutment remains the most vulnerable component under these conditions. The results demonstrate a steep rise in stress levels across the entire structure, particularly in the prosthetic components, under this high-tilt/high-angulation combination.

Conclusion:

Model 8 highlights a critical biomechanical concern. Although the cortical and cancellous bone-maintained stress within acceptable thresholds, the **abutment experienced excessive stress**, potentially risking component deformation or fracture. Likewise, elevated stress within

the implant structure further confirms that using a **35° abutment on a 30° tilted implant** results in **poor stress distribution and heightened mechanical load**, primarily at the implant-abutment interface. This model emphasizes the **importance of reducing angulation mismatch** to optimize load transmission and enhance the longevity of the implant-supported prosthesis.

The table no 11 enables a model-wise evaluation of biomechanical performance, highlighting stress concentration trends that influence implant longevity and prosthetic stability. Values marked with ⬆️ represent the **highest stress** recorded among all models for a given component, indicating zones of potential mechanical vulnerability. Conversely, values marked with ⬇️ denote the **lowest stress**, suggesting biomechanically favorable conditions.

The combined stress analysis table presents a comprehensive comparison of Von Mises stress values observed across all eight finite element models evaluating various implant tilt and transmucosal abutment angulation configurations. Each model includes maximum and minimum stress values recorded in the cortical bone, cancellous bone, abutment, and implant components under a 200 N vertical load. The table highlights the biomechanical performance of each configuration by identifying the peak and lowest stress values among all models. Notably, Model 8 exhibited the highest stress concentration in the abutment (1072.8 MPa ⬆️), indicating critical mechanical overload, while Model 1 showed the lowest abutment stress (216.92 MPa ⬇️), reflecting optimal biomechanical harmony due to matching angulations. Similarly, Model 7 recorded the highest cortical bone stress (33.162 MPa ⬆️), whereas Model 4 demonstrated the lowest (22.47 MPa ⬇️), suggesting reduced strain with near-aligned components. These trends emphasize the clinical relevance of minimizing angular discrepancies between implants and abutments to achieve favorable stress distribution, reduce mechanical complications, and enhance long-term success of implant-supported prostheses.

Table11- Combined Stress Analysis Table (Models 1 to 8)

Model	Cortical Bone Max	Cortical Bone Min	Cancellous Bone Max	Cancellous Bone Min	Abutment Max	Abutment Min	Implant Max	Implant Min
Model 1	26.636	0.0 ↓	4.2652	0.0 ↓	216.92 ↓	0.411	123.17 ↓	0.902
Model 2	29.484	8.25 ↑	3.4602	8.9217e-05 ↑	544.84	0.70236	235.01	1.1728 ↑
Model 3	24.007	0.0 ↓	5.5897	0.0 ↓	234.52	0.17	131.98	0.784
Model 4	22.47	0.0 ↓	7.128 ↑	0.0 ↓	506.73	0.319	205.73	1.054
Model 5	30.433	0.0 ↓	6.4	0.0 ↓	467.8	0.299	139.66	0.296 ↓
Model 6	22.098 ↓	0.0 ↓	5.378	0.0 ↓	773.16	1.474	276.05 ↑	0.441
Model 7	33.162 ↑	0.0 ↓	4.848	0.0 ↓	442.29	0.13 ↓	225.42	0.332
Modell 8	24.256	0.0 ↓	3.2516 ↓	0.0 ↓	1072.8 ↑	1.672 ↑	273.05	0.377

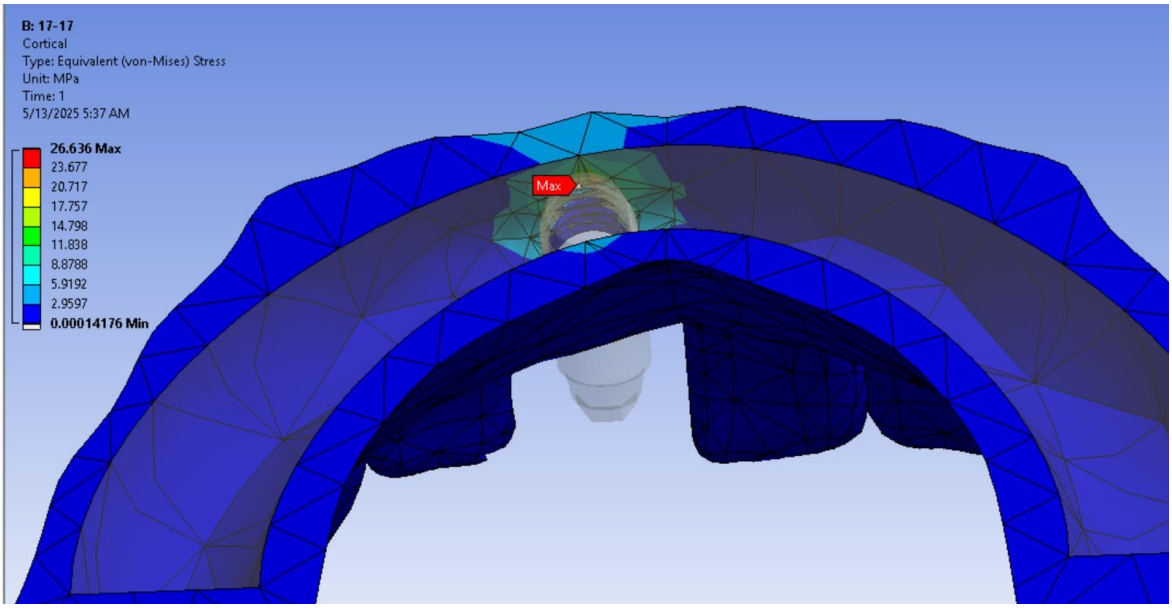


Fig 6(a) Represents stress transmitted on cortical bone- MODEL-1

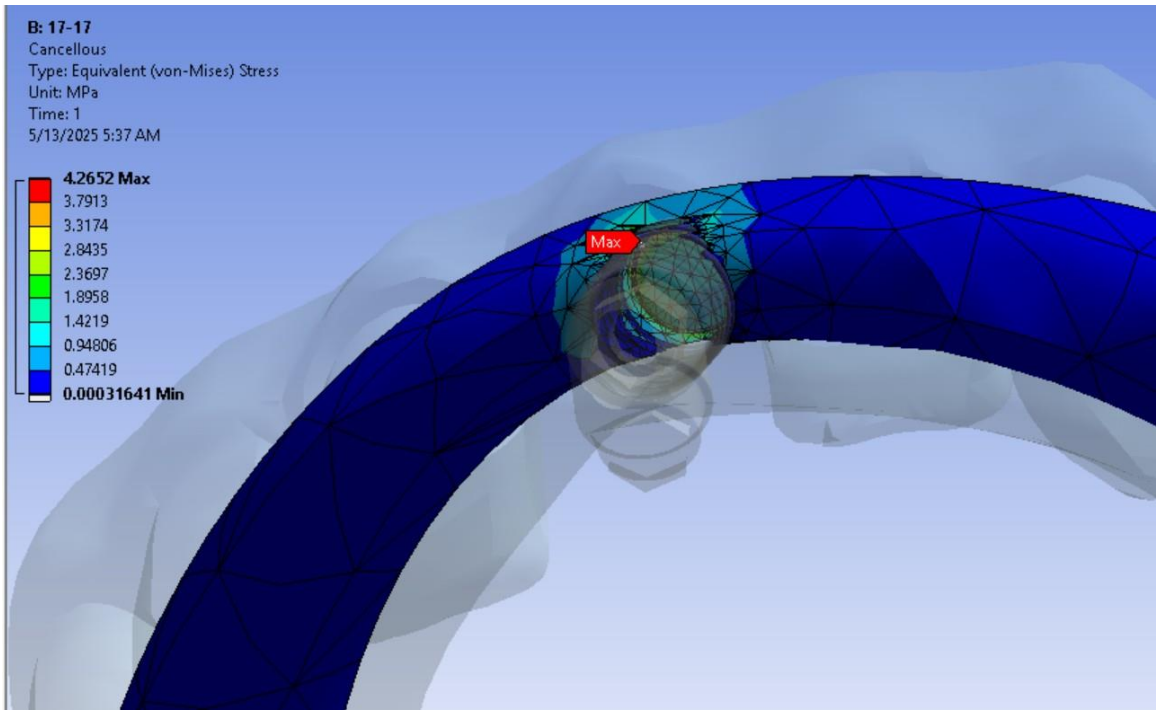


Fig 6(b) Represents stress transmitted on cancellous bone- MODEL-1

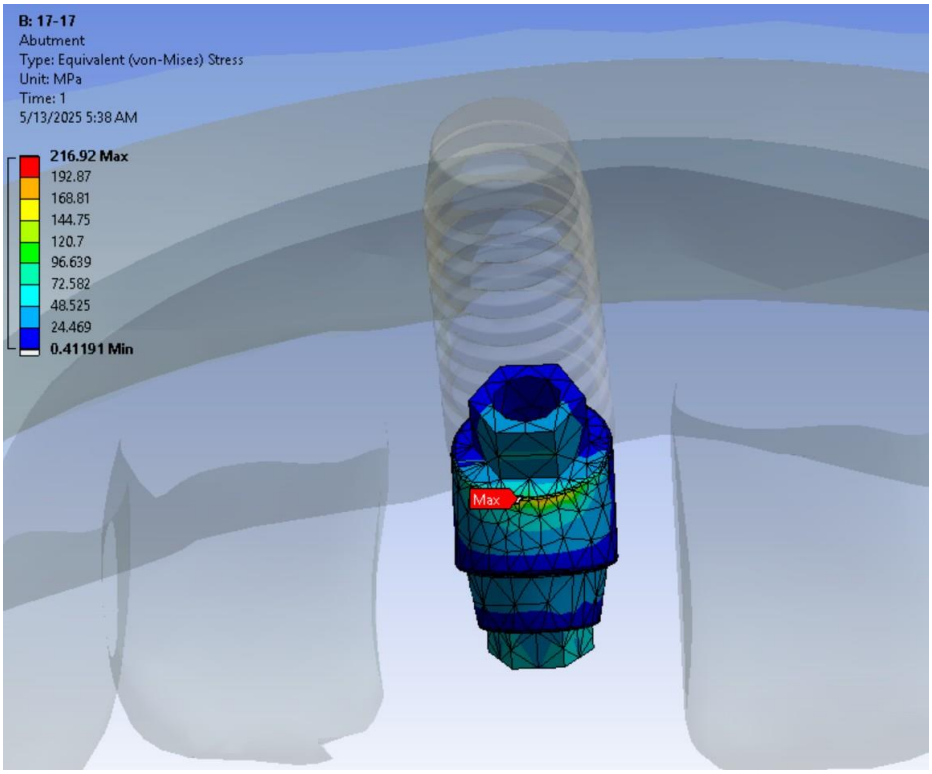


Fig 6(c) Represents stress transmitted on abutment- MODEL-1

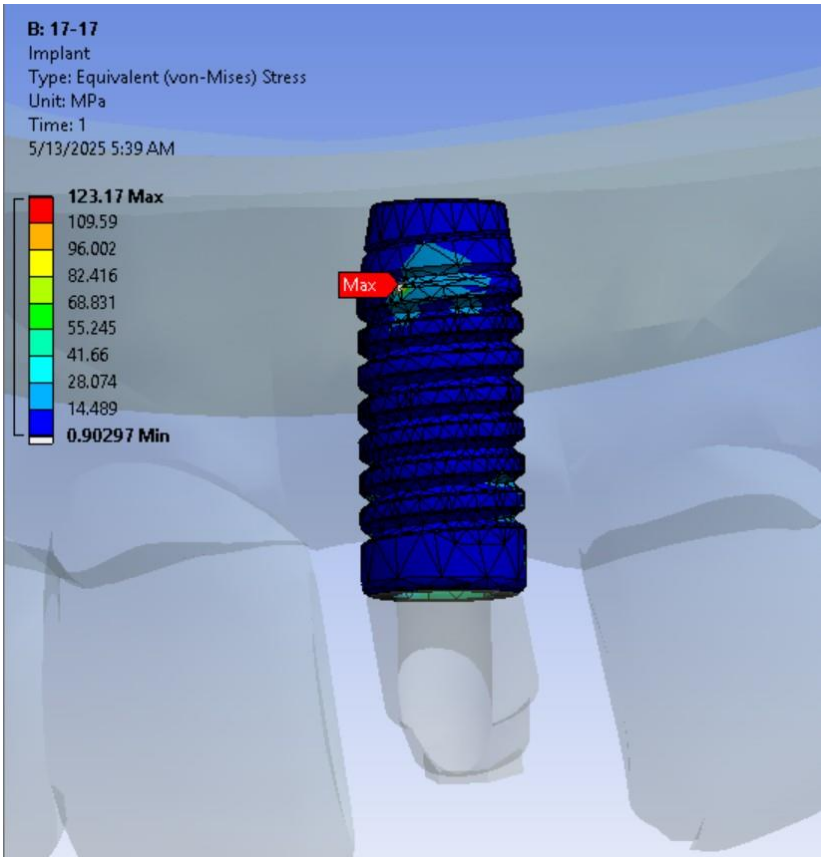


Fig 6(d) Represents stress transmitted on implant- MODEL-1

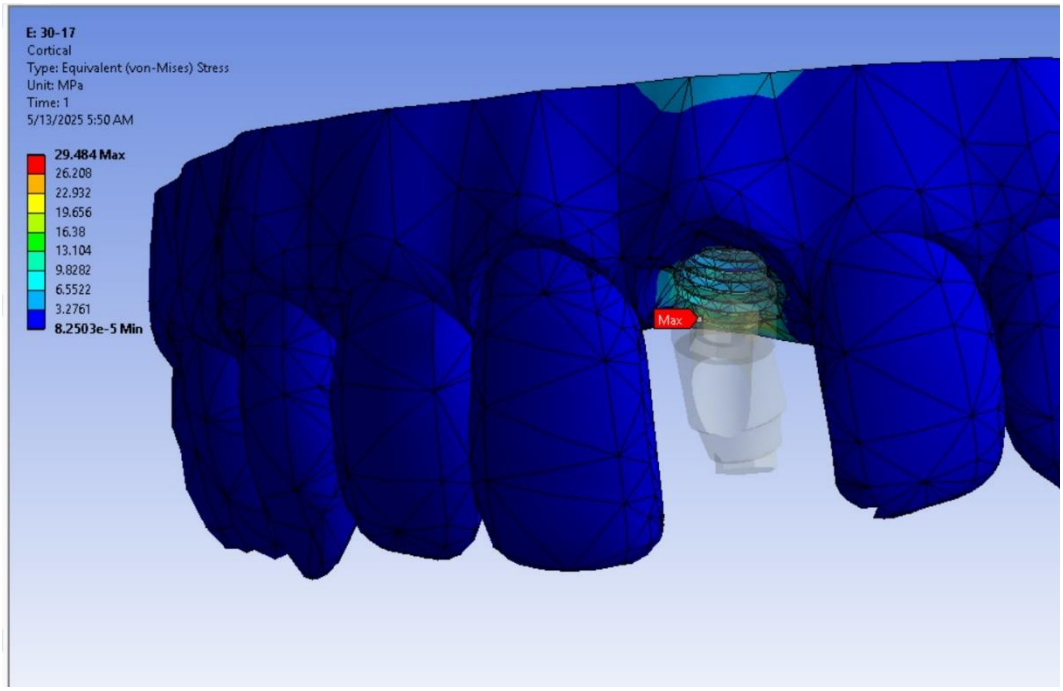


Fig 7(a) Represents stress transmitted on cortical bone- MODEL-2

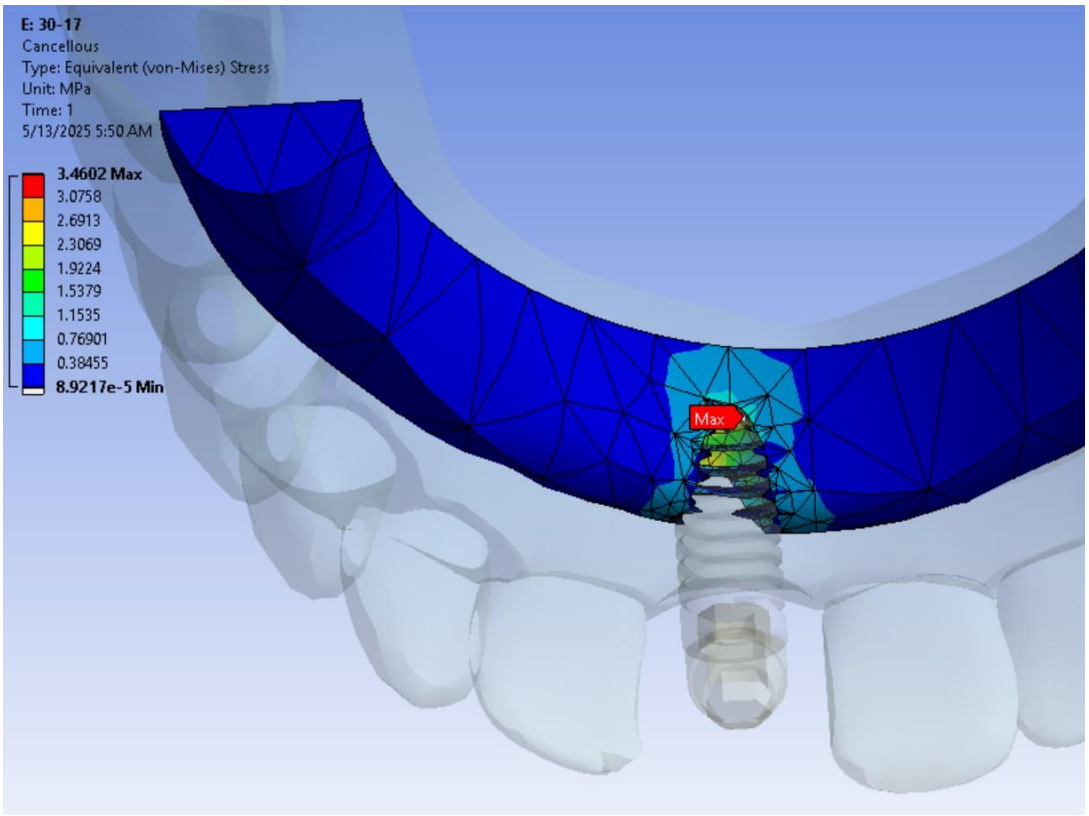


Fig 7(b) Represents stress transmitted on cancellous bone- MODEL-2

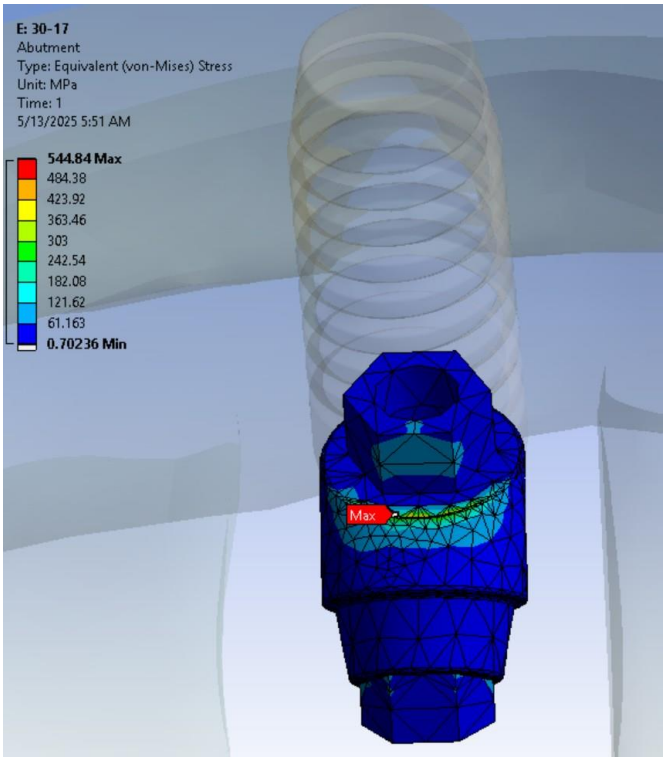


Fig 7(c) Represents stress transmitted on abutment - MODEL-2

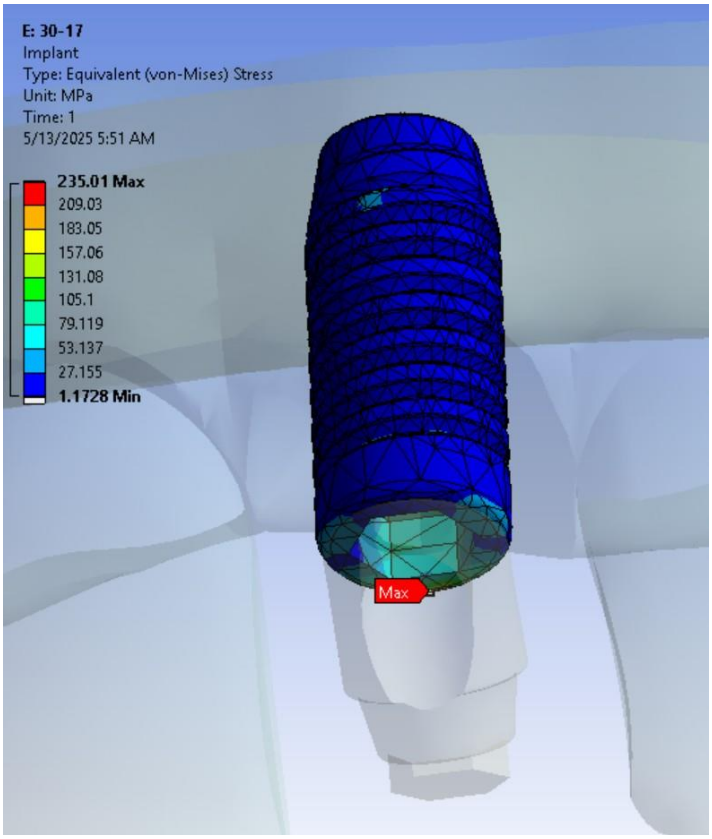


Fig 7(d) Represents stress transmitted on implant- MODEL-2

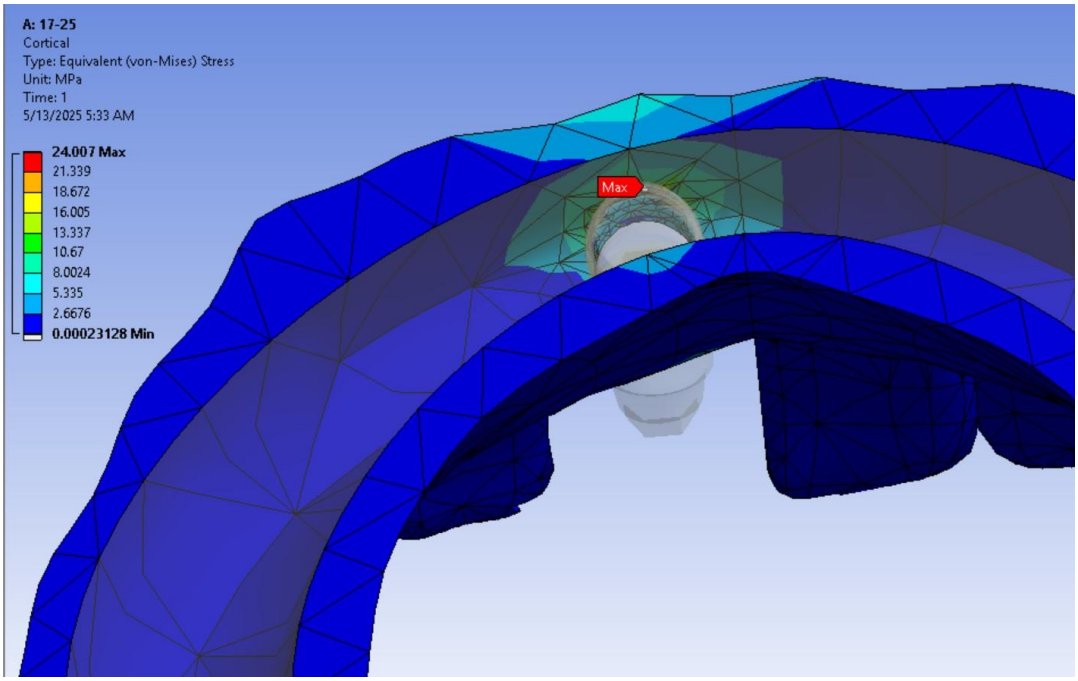


Fig 8(a) Represents stress transmitted on cortical bone- MODEL-3

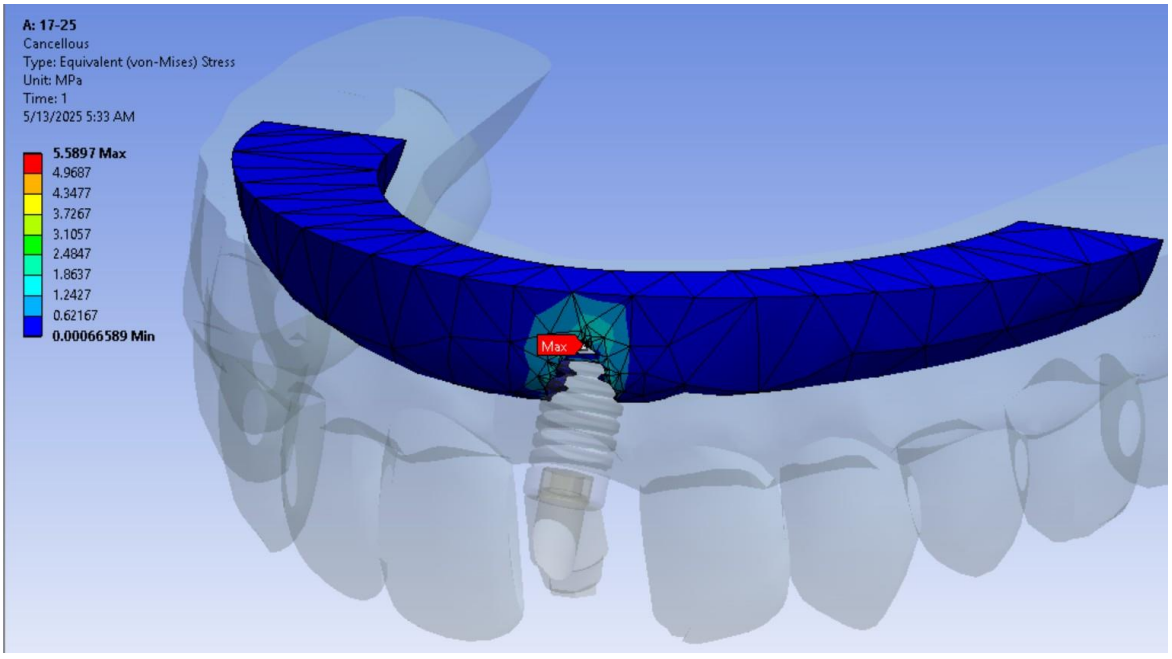


Fig 8(b) Represents stress transmitted on cancellous bone - MODEL-3

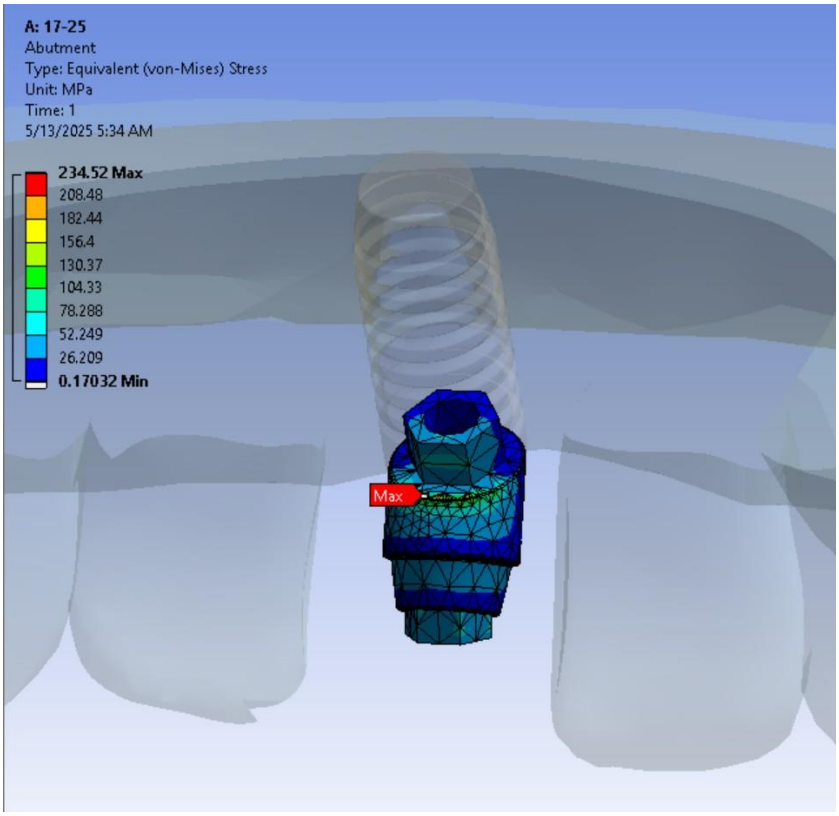


Fig 8(c) Represents stress transmitted on abutment- MODEL-3

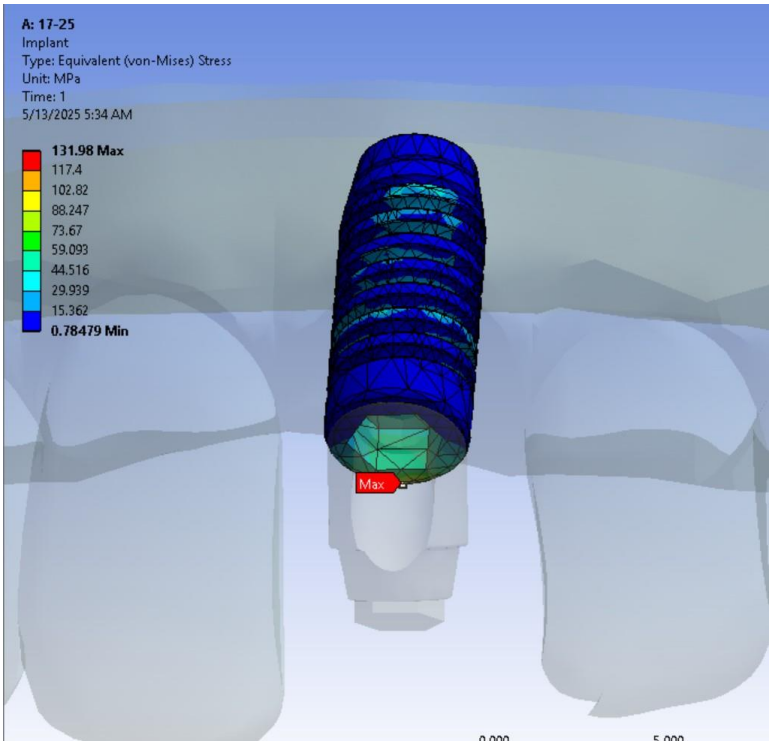


Fig 8(d) Represents stress transmitted on implant- MODEL-3

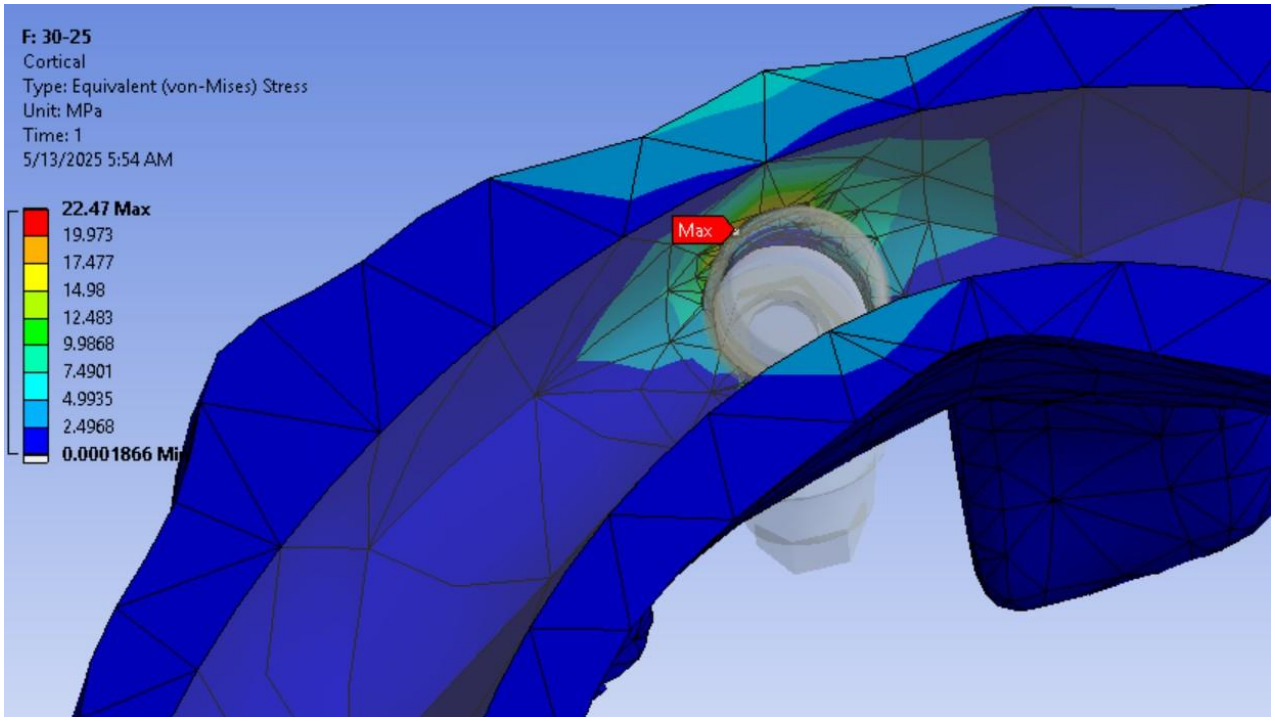


Fig 9(a) Represents stress transmitted on cortical bone- MODEL-4

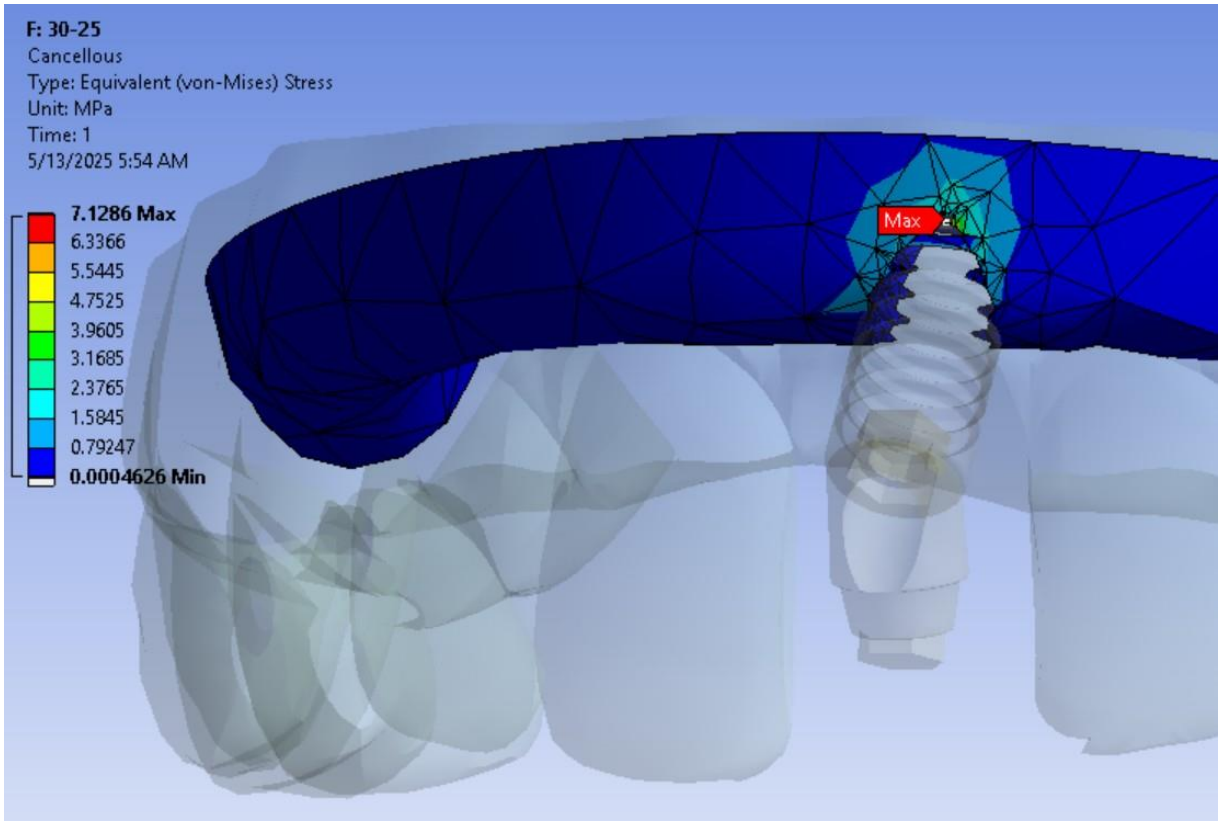


Fig 9(b) Represents stress transmitted on cancellous bone - MODEL-4

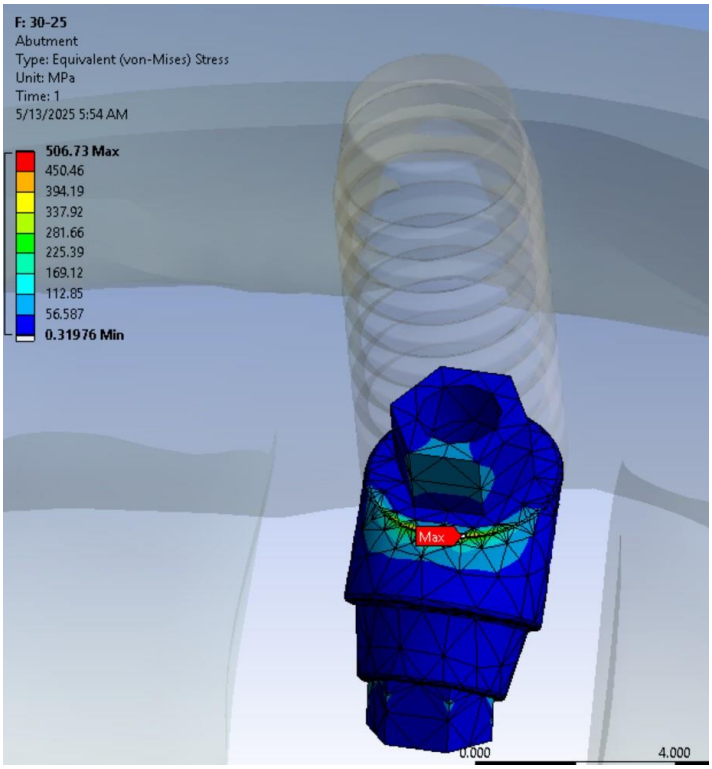


Fig 9(c) Represents stress transmitted on abutment- MODEL-4

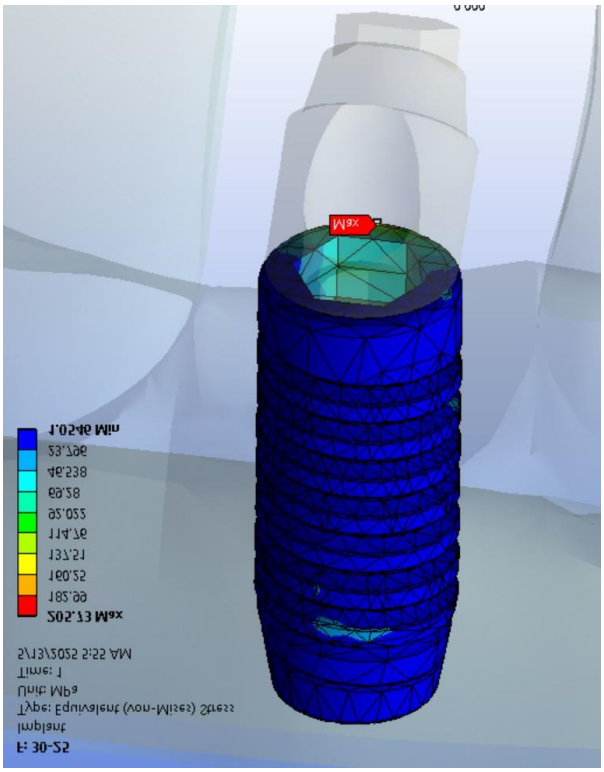


Fig 9(d) Represents stress transmitted on implant- MODEL-4

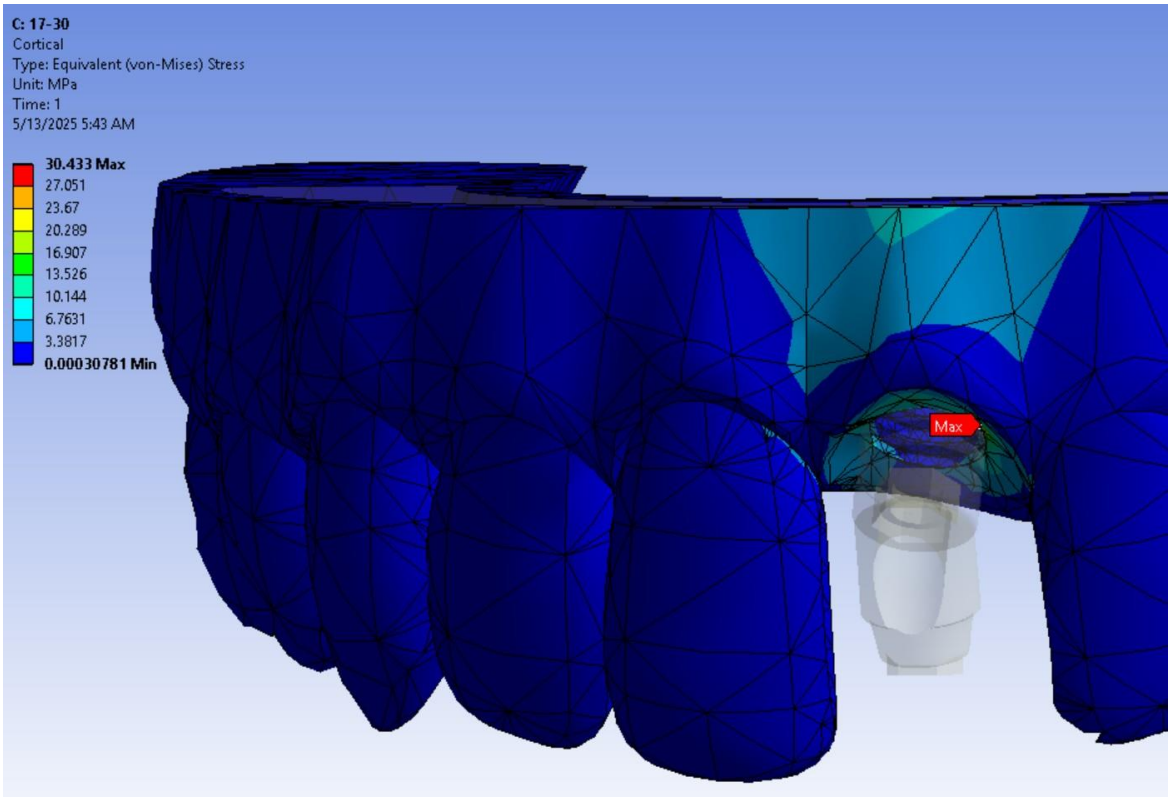


Fig 10(a) Represents stress transmitted on cortical bone- MODEL-5

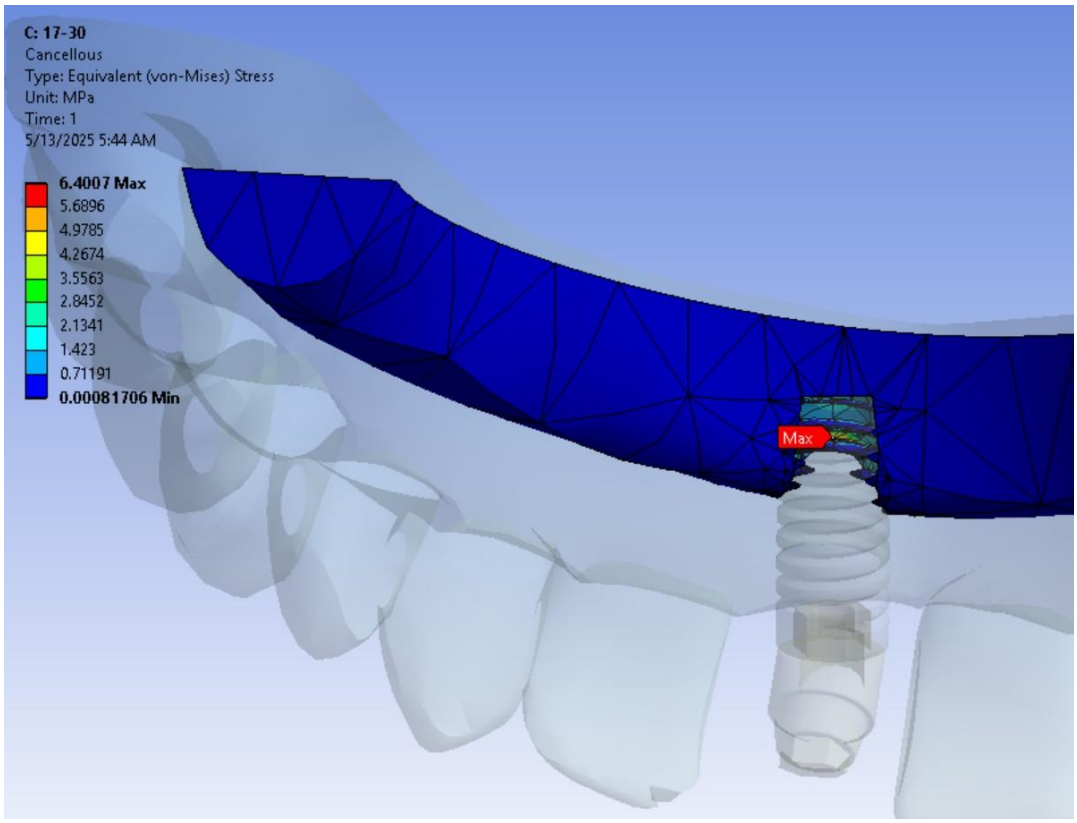


Fig 10(b) Represents stress transmitted on cancellous bone - MODEL-5

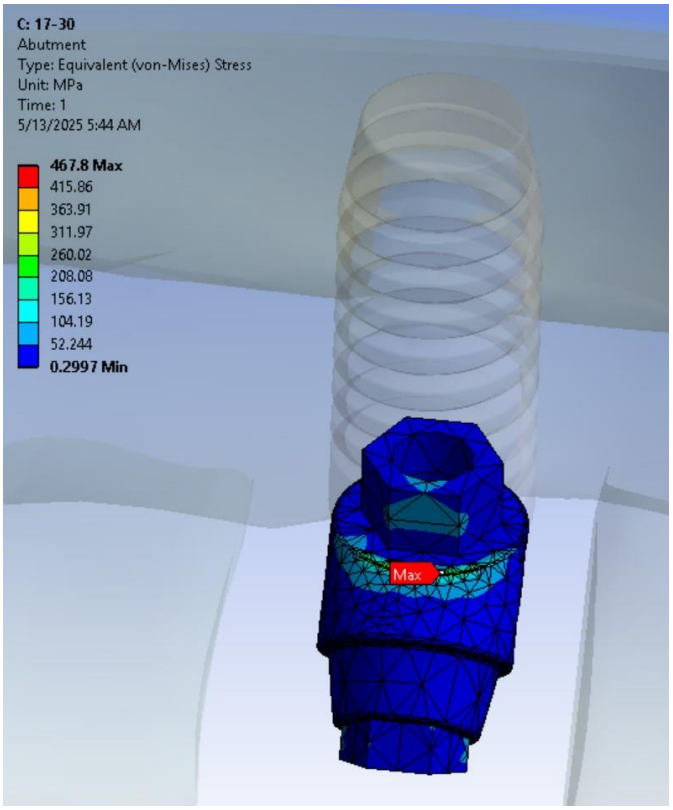


Fig 10(c) Represents stress transmitted on abutment- MODEL-5

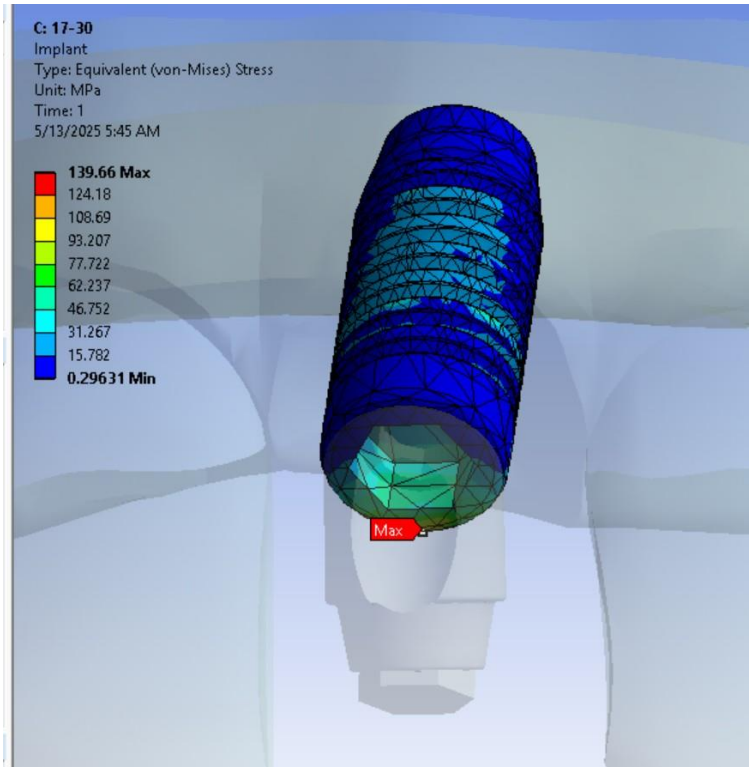


Fig 10(d) Represents stress transmitted on implant- MODEL-5

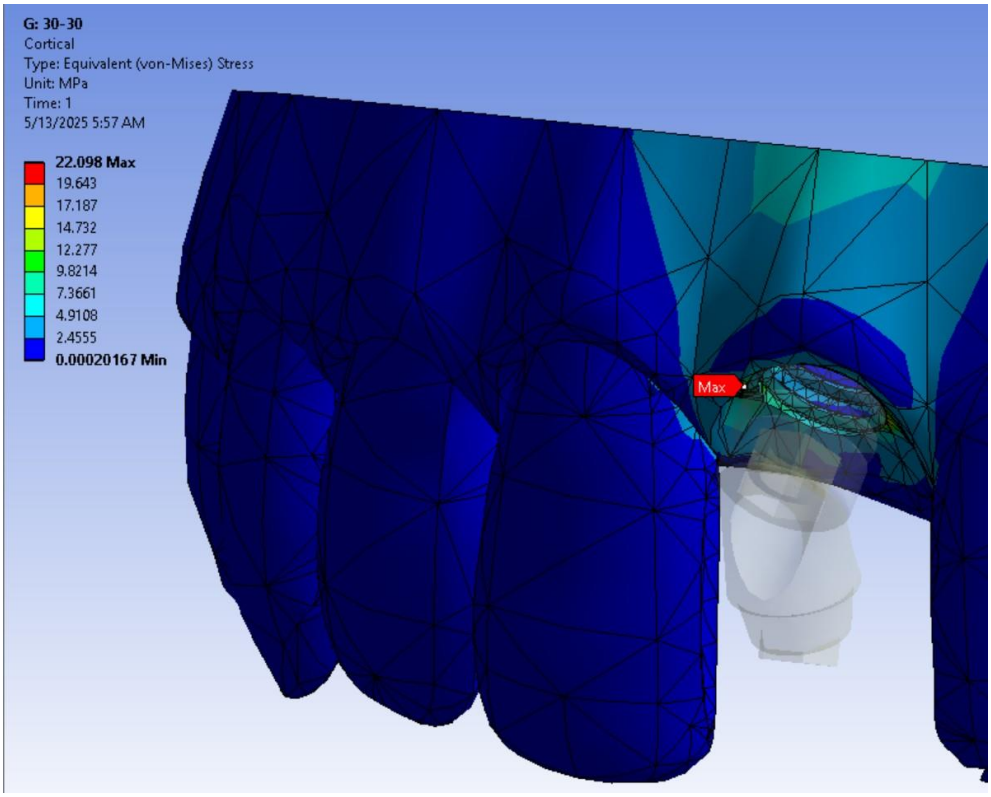


Fig 11(a) Represents stress transmitted on cortical bone- MODEL-6

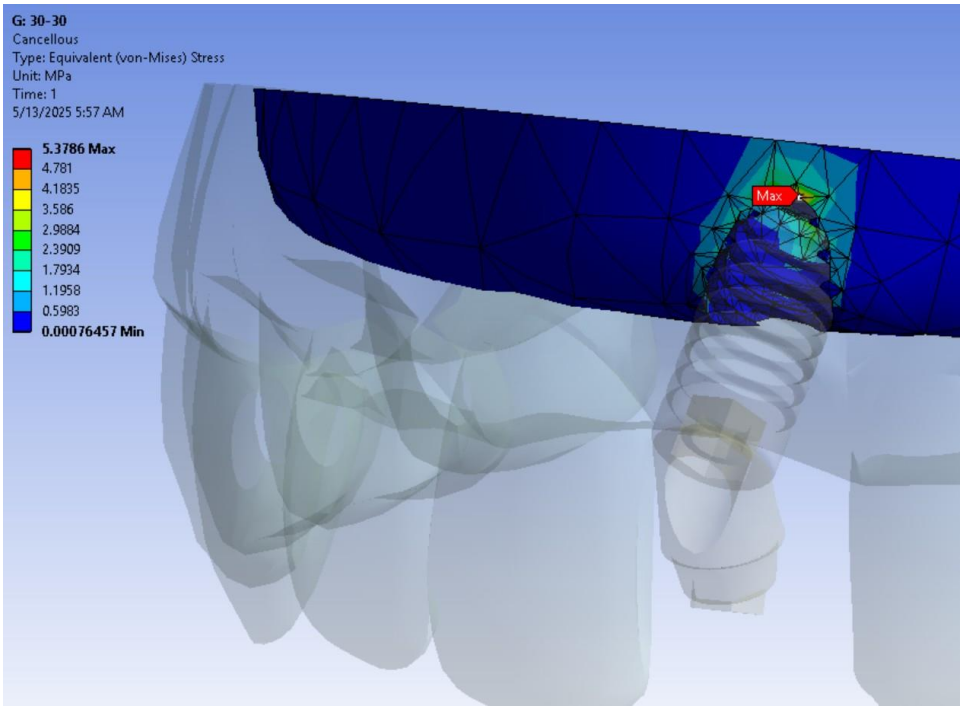


Fig 11(b) Represents stress transmitted on cancellous bone - MODEL-6

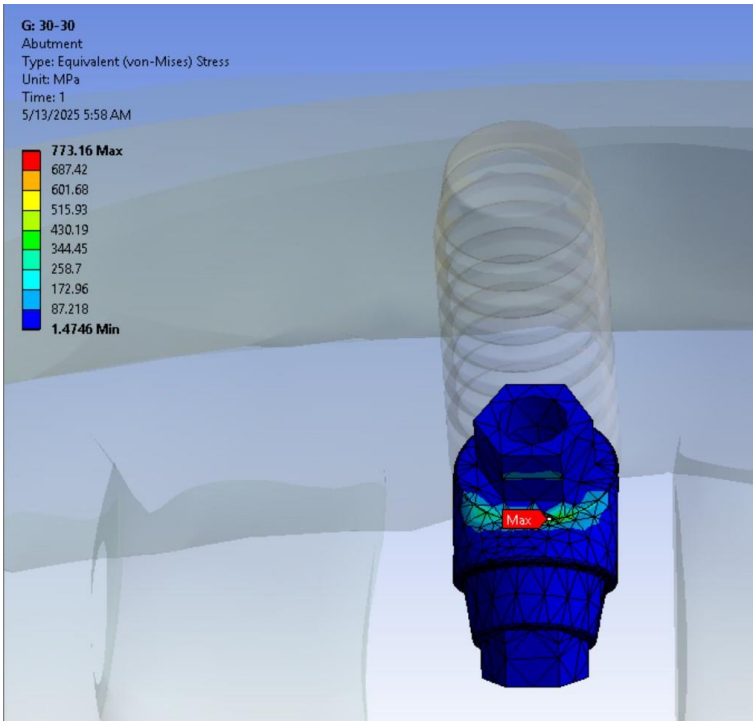


Fig 11(c) Represents stress transmitted on abutment- MODEL-6

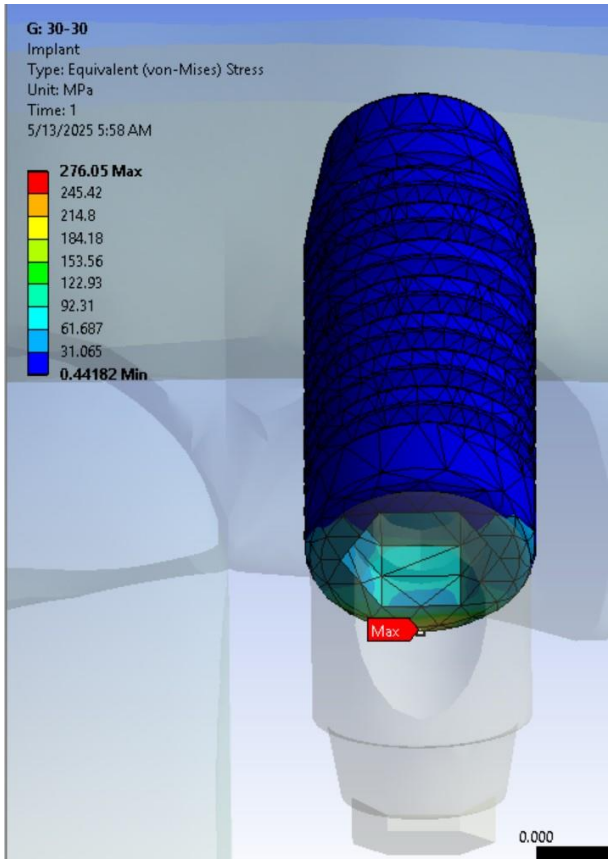


Fig 11(d) Represents stress transmitted on implant- MODEL-6

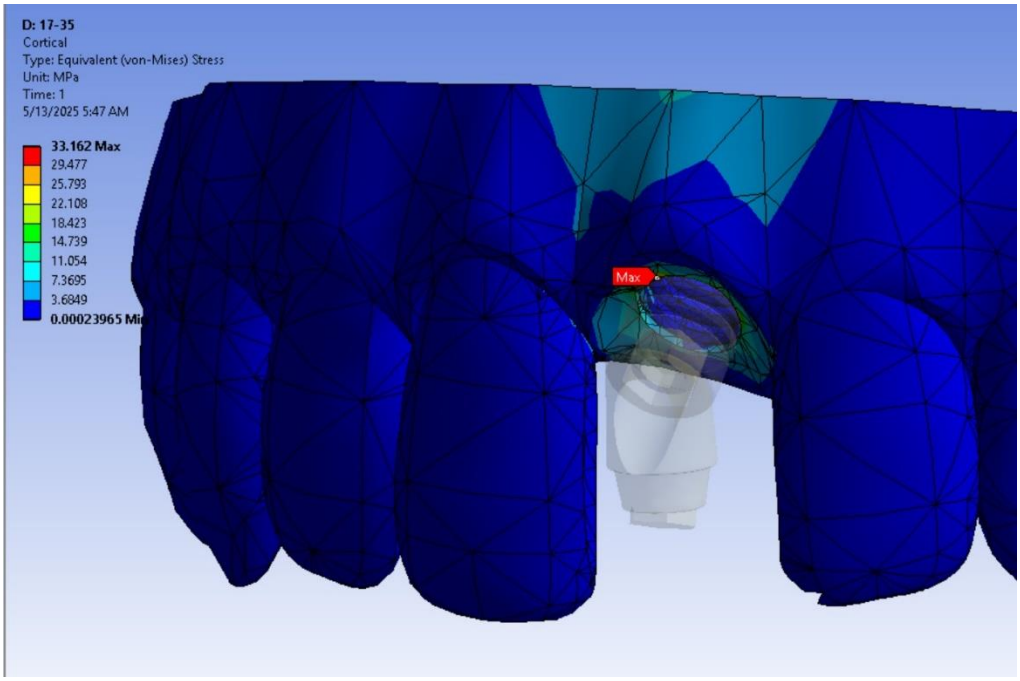


Fig 12(a) Represents stress transmitted on cortical bone- MODEL-7

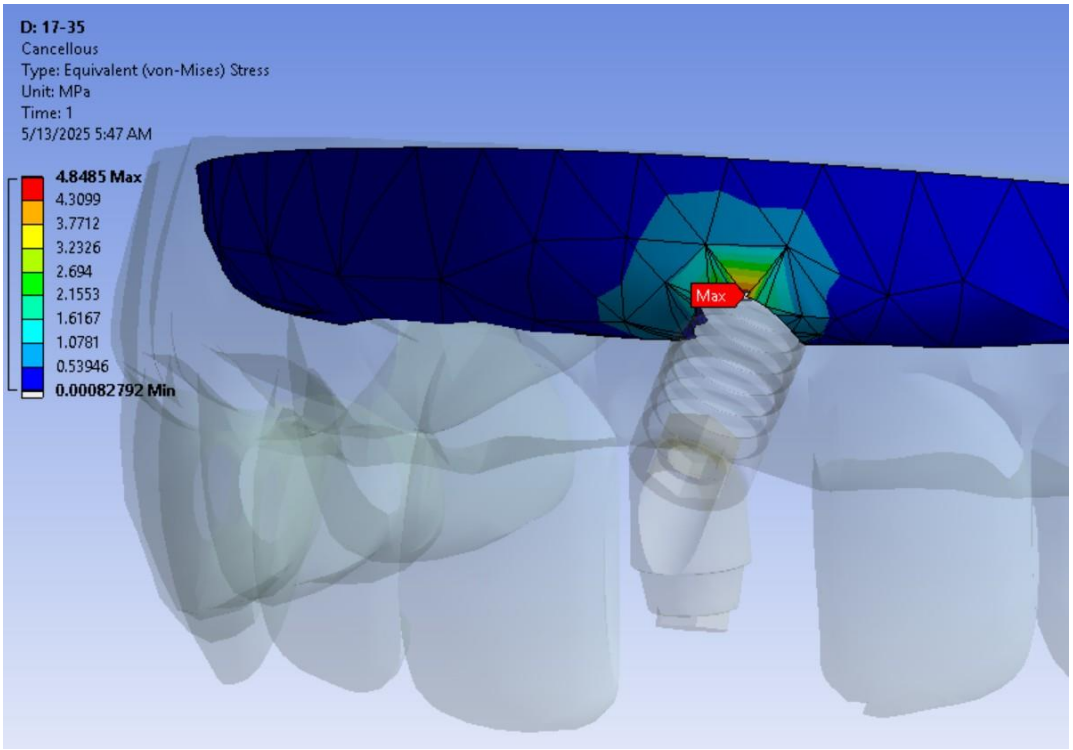


Fig 12(b) Represents stress transmitted on cancellous bone - MODEL-7

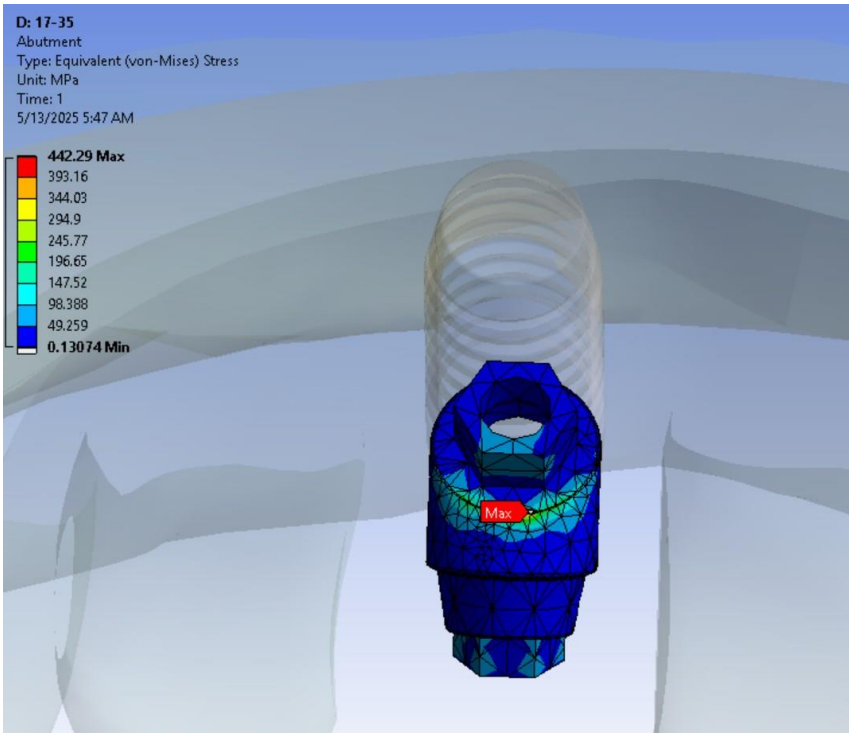


Fig 12(c) Represents stress transmitted on abutment- MODEL-7

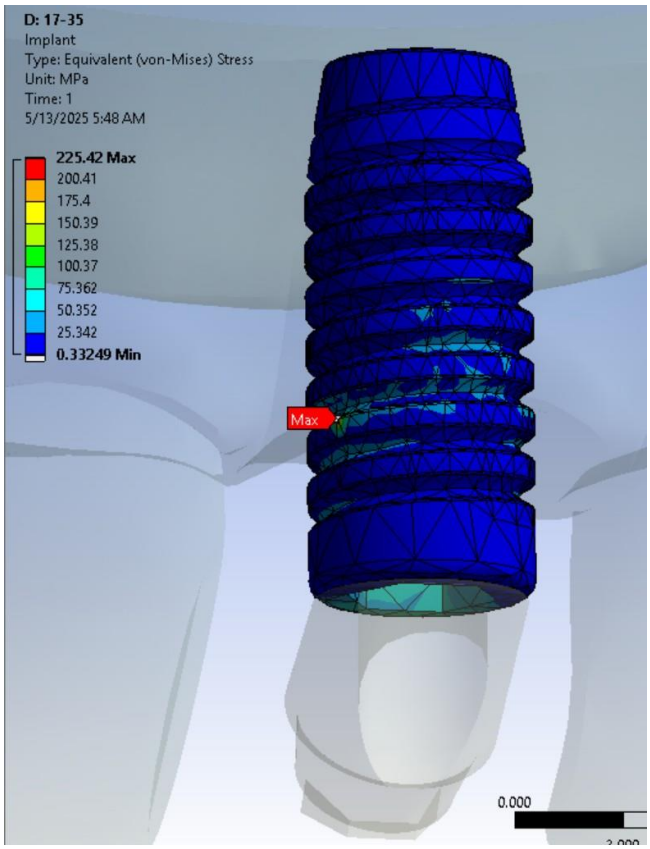


Fig 12(d) Represents stress transmitted on implant- MODEL-7

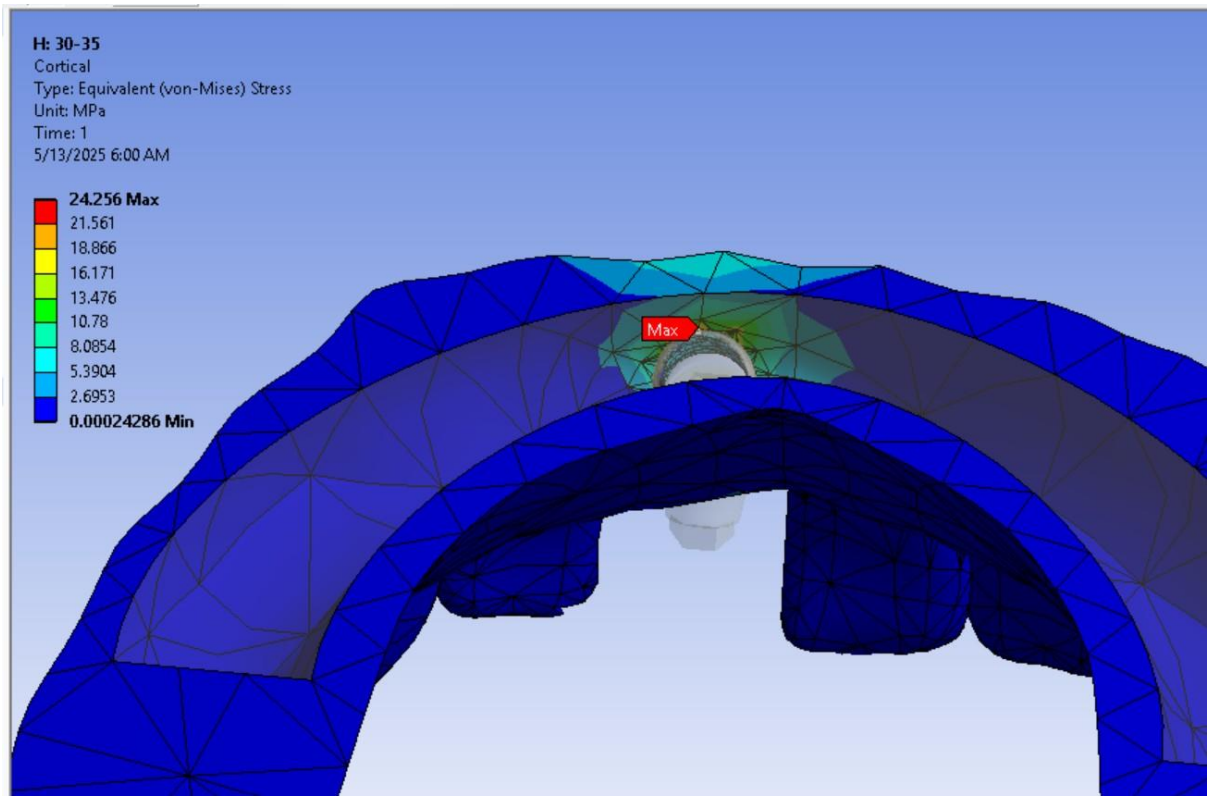


Fig 13(a) Represents stress transmitted on cortical bone- MODEL-8

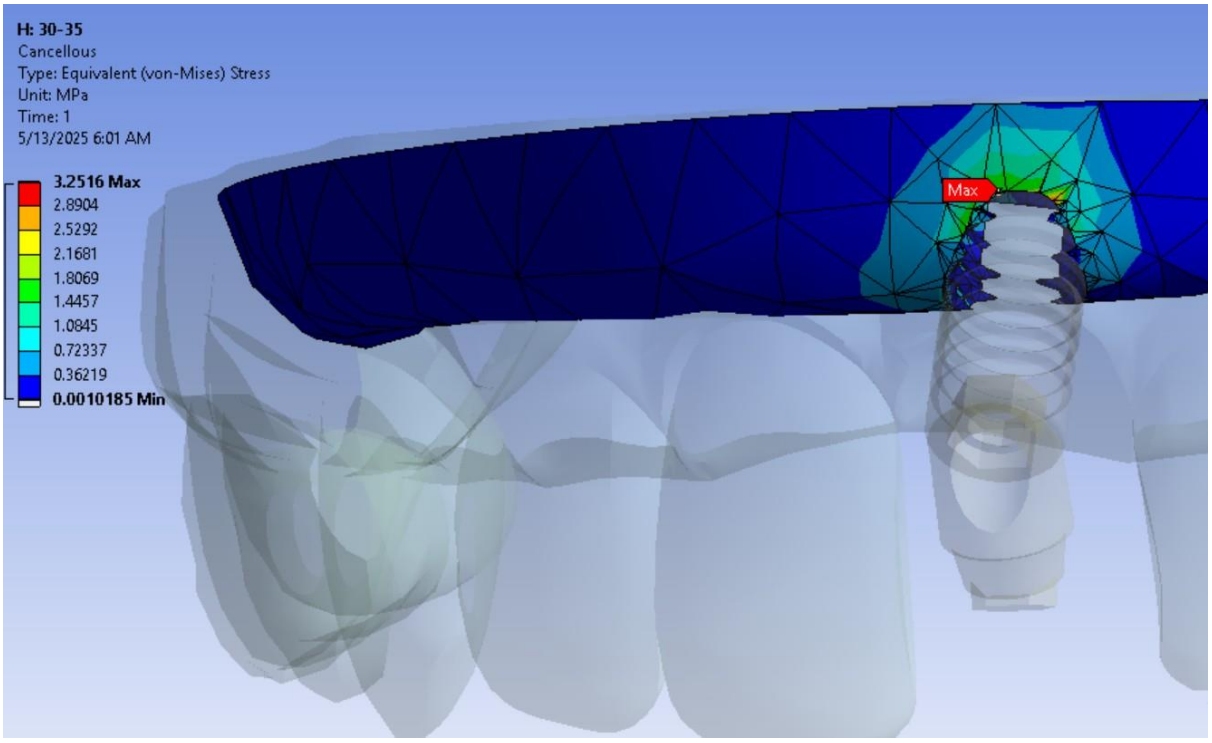


Fig 13(b) Represents stress transmitted on cancellous bone - MODEL-8

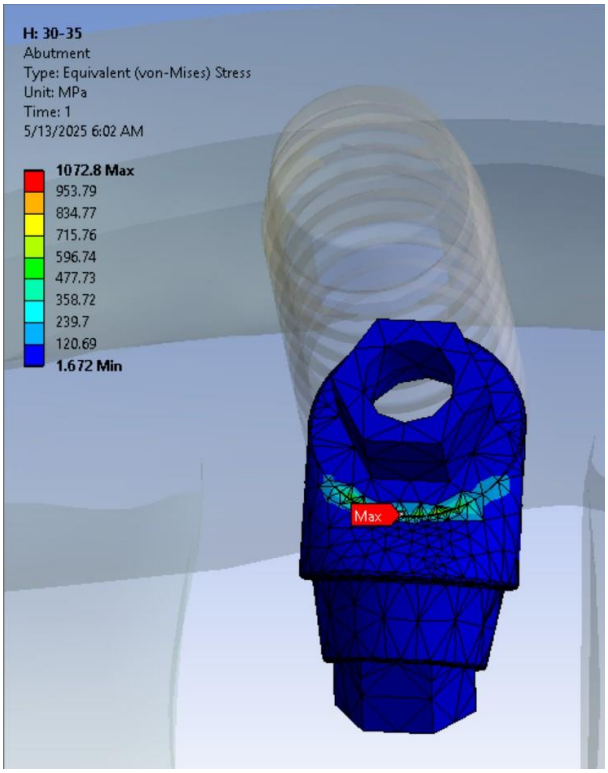


Fig 13(c) Represents stress transmitted on abutment- MODEL-8

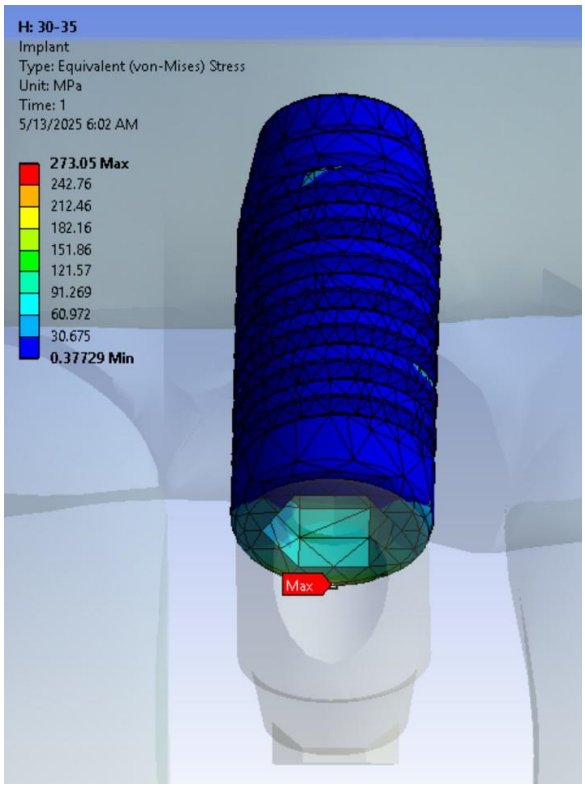


Fig 13(d) Represents stress transmitted on implant- MODEL-8

Discussion

DISCUSSION

The present in-vitro finite element method (FEM) study was undertaken to examine the biomechanical consequences of varying implant tilt angles and transmucosal abutment angulations on the stress distribution in the peri-implant region. This research is grounded in the increasing use of angulated implants to overcome anatomical limitations and the accompanying need to optimize prosthetic components for mechanical harmony. By analyzing eight distinct models simulating implant placements at angulations of 17°, 25°, 30°, and 35° combined with either 17° or 30° transmucosal abutments, this study elucidates the impact of angular relationships on cortical bone, cancellous bone, abutments, and implant bodies using ANSYS R17.[4,7,8,11,15,17]

Stress analysis in implant-supported prostheses is crucial due to its direct implication on osseointegration, prosthetic stability, and component longevity. Numerous studies have shown that biomechanical mismatch between implant placement and abutment angulation can result in off-axis loading, leading to crestal bone loss, screw loosening, and eventual implant failure (Weinberg LA, 1993; Rangert et al., 1997; Geng et al., 2001). Axially aligned implants tend to transfer forces more uniformly, preserving both prosthetic and biological components. In contrast, angular discrepancies introduce complex stress trajectories that concentrate forces at biomechanically sensitive interfaces such as the implant-abutment junction and crestal bone.

In this study, CBCT-derived 3D models were used to create anatomically accurate maxillary simulations. Implants were modeled as 3.75 × 11.5 mm titanium screws (Adin), and abutments were either 17° or 30° angulated TMAs. Material properties were assigned using values from **Sevimay et al. (2005) and Turker et al. (2007),** assuming isotropy and homogeneity. Static loads of 200 N were applied vertically, a standard representation of occlusal loading in FEM

studies. All implants were assumed to be fully Osseo integrated with a perfect bone-implant interface **(Misch et al., 2005).**

The finite element analysis across all implant-abutment angulation combinations revealed clear biomechanical trends influenced by both implant tilt and abutment alignment. When the **implant and abutment were aligned at 17°,** the stress distribution remained uniform and well within physiological limits across all structures. This condition facilitated effective axial load transmission, reducing torsional forces and minimizing biomechanical strain. Such favorable conditions align with the findings of **Weinberg** [1], who highlighted the importance of axial loading in preventing stress concentration and promoting long-term implant stability.[1,2,4,5,7,8]

In contrast, introducing a **13° mismatch** between a 17° implant and 30° abutment caused a sharp increase in abutment stress (544.84 MPa) and implant stress (235.01 MPa), indicating torsional overload and inefficient load transfer. These outcomes support observations by **Anitua et al.** [2], who associated mismatched angulations with torque-related complications and screw loosening.[1,2,4,9,10,13,15]

An **8° mismatch** between a 25° implant and 17° abutment generated moderate stress elevation, while even a 5° mismatch between a 25° implant and 30° abutment produced abutment stress levels as high as 506.73 MPa. These results affirm conclusions by **Ozan et al.** [3], who noted that small angular mismatches can still create significant biomechanical disturbances, especially when compounded by implant inclination.[1,2,3,4,15]

A **30° implant restored with a 17° abutment** (13° mismatch) also demonstrated pronounced stress concentrations, particularly at the abutment-implant interface, which aligns with **Geng et al.** [4], who reported that oblique loading due to angular discrepancy exacerbates stress in both cortical bone and prosthetic components. Interestingly, even with **angular harmony at 30°,** implant and abutment stress peaked at 276.05 MPa and 773.16 MPa, respectively,

highlighting that **implant tilt alone** is a critical factor in stress amplification. This confirms the findings of **Tribst et al**. [5] , who showed that increased implant inclination inherently elevates mechanical loading on implant-abutment complexes.[1,2,4,5,7,8,11,15]

A **35° implant restored with a 17° abutment** (18° mismatch) resulted in extremely high stress values at the crestal bone and abutment neck, nearing physiological thresholds. This is consistent with **Ozan et al**. [3], who reported that steep angulation mismatches lead to critical biomechanical overload, jeopardizing peri-implant bone health. Furthermore, when a **35° implant was restored with a 30° abutment** (5° mismatch), the abutment stress peaked at **1072.8 MPa**, the highest recorded in the study. This finding illustrates that even minimal mismatch, when combined with steep implant tilt, demands advanced prosthetic planning and material reinforcement to prevent failure under cyclic functional loads.[1,2,15]

Among all the configurations analyzed, Model 1, featuring perfect axial alignment between the implant and abutment, consistently demonstrated the most favorable biomechanical outcomes. This precise angulation match facilitated efficient axial force transmission, minimizing stress concentrations across all components. In contrast, models exhibiting an **angular mismatch greater than 5°**, such as Models 2, 5, and 7, showed a marked escalation in stress, particularly at the crestal bone and abutment-implant interface. These mismatches introduced off-axis loading, resulting in torsional and bending forces that compromised mechanical stability. Across all models, the **abutment emerged as the most stressed component**, reaffirming its role as the mechanical weak link within the implant system—a finding supported by **Tribst et al. (2018)**. Furthermore, it was observed that **higher implant tilts inherently increase overall stress**, even in cases with angulation-matched abutments as seen in Models 6 and 8, indicating that implant inclination alone significantly amplifies biomechanical demands. The study establishes a clear biomechanical relationship between implant-abutment angulation compatibility and stress distribution. While matched angulations ensure better load

distribution, high implant tilts, even when matched, demand attention to prosthetic reinforcement. Future research should explore dynamic loading, patient-specific variability, and fatigue testing to complement FEA findings.[1,2,3,4,5]

Analyzing stress distribution in cortical bone in model 1 (Implant 17° with Abutment 17°) In this perfectly aligned configuration, the maximum **cortical bone stress recorded was 26.636 MPa**, primarily located at the **crestal region near the implant neck**. The axial load transfer allowed for uniform stress dissipation, indicating a **biomechanically favorable setup**. The stress remained **within physiological limits**, with no signs of overload, highlighting the benefits of angulation harmony. [7,8,11]

Model 2 (Implant 17° with Abutment 30°) Model 2 showed a **peak cortical bone stress of 29.484 MPa**. The **13° mismatch** led to **stress intensification at the labial aspect of the crestal bone**, where the implant-abutment interface bore significant bending forces. This stress concentration raises concern for **marginal bone resorption**, especially under cyclic occlusal loading. [1,2,4,5,7,8,11]

Model 3 (Implant 25° with Abutment 17°) In Model 3, **the maximum stress in cortical bone was 24.007** MPa. Although the mismatch was **moderate (8°),** it produced **asymmetric stress distribution**. Concentrated stress at the crestal shoulder suggested that even small mismatches can influence load paths, potentially leading to bone remodeling or microfractures over time. [7,8,11]

Model 4 (Implant 25° with Abutment 30°) Despite only a **5° mismatch**, Model 4 exhibited 22.47 MPa maximum **stress in the cortical bone**. The stress was localized around the **crestal shoulder and labial side**, suggesting that **slight angular variations still impact cortical stress distribution**, especially in regions with thinner cortical plates.[7,8,11]

Model 5 (Implant 30° with Abutment 17°) This model produced the **highest cortical stress value of 30.433 MPa**. The **13° angular discrepancy** created significant **bending moments**, with stress peaking at the labial crestal region. This scenario highlights the **biomechanical risk of severe angulation mismatch**, which could compromise **bone health and implant longevity.**

Model 6 (Implant 30° with Abutment 30°) Angular harmony helped control stress levels to **22.098 MPa** despite the steep implant tilt. Stress distribution was relatively **uniform across the crestal bone**, suggesting that **matching angulations effectively redirects occlusal loads**, even in high-tilt conditions. This setup illustrates how **prosthetic alignment can compensate for implant orientation.**

Model 7 (Implant 35° with Abutment 17°) Model 7 showed a **peak cortical bone stress of 33.162 MPa, the highest in the entire study**. The **18° mismatch** generated intense stress at the **labial crestal margin**, pointing to the **critical biomechanical instability** of high-mismatch configurations. Such stress levels are associated with **marginal bone breakdown and early peri-implant complications**.[7,8,11]

Model 8 (Implant 35° with Abutment 30°) In this configuration, **a 5° mismatch** yielded a **maximum cortical bone stress of 24.256 MPa.** Although better than Model 7, the **labial crestal** region still showed moderate stress accumulation. The results suggest partial **biomechanical compensation,** but the high implant tilt continues to influence cortical stress patterns.[7,8,11]

Across all eight models, **cortical bone stress was consistently highest at the crestal region**, especially on the **labial aspect**. Model 1, with perfect angulation match, showed the most **favorable stress profile**, while Models 5 and 7, with 13° and 18° mismatches respectively, recorded the **highest stress values**. Even minor mismatches of 5° (Models 4 and 8) caused

localized stress intensification. These findings reaffirm the importance of **implant-abutment angulation compatibility** in minimizing crestal bone overload and enhancing long-term implant success.[7,8,11]

Analyzing stress distribution in cancellous bone in model 1 (Implant 17° with Abutment 17°) In Model 1, **the cancellous bone exhibited a maximum stress of 4.2652 MPa**, which was evenly distributed around the cervical portion of the bone surrounding the implant. The alignment between implant and abutment ensured axial load transfer, resulting in **low, physiologically acceptable stress levels**. This uniform distribution signifies **minimal risk for trabecular bone remodeling or microdamage,** reinforcing the biomechanical advantage of angulation harmony. [4,15,17]

Model 2 (Implant 17° with Abutment 30°) A 13° mismatch in this model led to a **slightly reduced cancellous bone stress of 3.4602 MPa**, though stress distribution became **less uniform**. Despite the relatively low stress magnitude, the angular mismatch induced **non-axial load transfer**, which caused **stress accumulation in isolated zones,** particularly at the cervical trabecular bone. This uneven stress pattern may **predispose the bone to long-term fatigue** under functional load.[1,2,3,4,15,17]

Model 3 (Implant 25° with Abutment 17°) Model 3 showed a **maximum stress of 5.5897 MPa in cancellous bone**, marking an increase over both Models 1 and 2. The angular mismatch of 8° caused **noticeable stress intensification near the cervical trabecular interface**, suggesting that **even moderate angular discrepancies** can elevate mechanical demand on cancellous bone. The porous structure of this bone type makes it especially susceptible to **microfractures and remodeling** when subjected to such loads.[1,2,3,4,15,17]

Model 4 (Implant 25° with Abutment 30°) Despite a relatively mild 5° mismatch, Model 4 **exhibited the highest cancellous bone stress in the study at 7.128 MPa**. This indicates that

even minor misalignments, when combined with moderate implant tilt, can cause **considerable biomechanical stress.** The stress was particularly concentrated at **the cervical and labial regions**, indicating a **greater risk for structural weakening and bone adaptation** over time. [4,15,17]

Model 5 (Implant 30° with Abutment 17°) In this configuration, the cancellous bone stress reached **6.4007 MPa**, which is significantly elevated and reflects **intense oblique force transmission** due to a 13° mismatch. The stress distribution pattern was broad and diffused into deeper bone areas, indicating that **load was not adequately redirected through axial pathways.** This could compromise **bone-implant contact** and increase the likelihood of **trabecular damage**. [4,15,17]

Model 6 (Implant 30° with Abutment 30°) Angular alignment helped reduce cancellous bone stress to **5.378 MPa**, even though the implant tilt remained steep. The stress was more symmetrically distributed, which **mitigated peak concentrations** seen in mismatched models. This supports the idea that **matching angulations can effectively dampen mechanical loads**, even at higher implant inclinations. [4,15,17]

Model 7 (Implant 35° with Abutment 17°) Despite the most extreme mismatch (18°), Model 7 **recorded a relatively low cancellous bone stress of 4.848 MPa.** However, this lower value does not imply biomechanical safety. The stress was **highly localized**, particularly at the **labial cervical zone**, where trabecular bone is thinnest. This could result in **focused fatigue and early microstructural changes** despite the moderate stress magnitude. [4,15,17]

Model 8 (Implant 35° with Abutment 30°) Model 8 displayed **the lowest cancellous bone stress** among the **higher-tilt models at 3.2516 MPa**, owing to better angular alignment (5° mismatch). The stress remained diffuse and relatively uniform, suggesting that stress **compensation was partially achieved**. Nevertheless, due to the steep implant tilt, attention

must still be paid to cumulative stress effects under occlusal function. The stress distribution in cancellous bone across the eight models reveals that **alignment between implant and abutment reduces peak stress** and promotes **uniform load sharing**. Models with **severe mismatches** (Models 2, 3, 5, 7) demonstrated **non-uniform stress concentration**, particularly near the cervical and labial zones. Notably, Model 4 **exhibited the highest cancellous bone stress (7.128 MPa),** despite only a 5° mismatch, emphasizing the sensitivity of cancellous bone to combined angular deviation and implant tilt. These findings underline the importance of **angulation control** and **custom abutment usage** to safeguard cancellous bone integrity.[1,2,4,5,15,17]

Analyzing stress distribution in abutment in model 1 (Implant 17° with Abutment 17°) In Model 1, the abutment experienced a **maximum von Mises stress of 216.92 MPa,** with a minimum of 0.411 MPa. The stress concentration was **primarily located at the junction** of the abutment with the implant fixture. Due to the **perfect alignment of angulations**, **the load was transmitted axially**, resulting in a **smooth and evenly distributed stress pattern.** This model represented the **most biomechanically favorable condition** for abutment performance, with minimal bending moments or torque and **reduced risk of mechanical complications** such as screw loosening or abutment fracture. [1,2,4,5]

Model 2 (Implant 17° with Abutment 30°) A 13° mismatch between implant and abutment in Model 2 caused the abutment to endure a **maximum stress of 544.84 MPa**—the **second highest** in the study. The sharp angular deviation introduced substantial shear **forces** at the implant-abutment interface. Stress localized around the **midsection of the abutment and the collar**, reflecting significant mechanical strain. This mismatch led to **increased torque and off-axis loading**, elevating the risk of **component deformation, screw loosening, and long-term fatigue failure.**

Model 3 (Implant 25° with Abutment 17°) Model 3 presented an **8° mismatch**, leading to a **maximum abutment stress of 234.52 MPa** and a minimum of 0.170 MPa. Although less severe than Model 2, the stress was again **localized at the implant-abutment interface**, indicating bending forces at play. This model illustrated how **even moderate mismatches introduce biomechanical inefficiency**, potentially affecting long-term prosthetic stability. [1,2,15]

Model 4 (Implant 25° with Abutment 30°) Despite only a **5° mismatch**, the abutment stress peaked at **506.73 MPa**, a surprisingly high value. The stress localized sharply at the abutment collar, suggesting that **partial angular compensation still leaves residual torque.** This level of stress indicates a high probability of **mechanical complications**, such as **thread stripping or fracture** under cyclic loading. This model emphasizes **that even slight angular mismatches can critically stress the abutment.** [1,2,3,4,15]

Model 5 (Implant 30° with Abutment 17°) This configuration showed a **maximum abutment stress of 467.8 MPa**, concentrated at the **implant-abutment interface.** The **13° mismatch** resulted in severe torque and **bending moments**, increasing the chances of **abutment fatigue**, **loosening, or fracture**. This model reinforces the clinical concern that **high angular mismatch significantly compromises the prosthetic connection.** [1,2,3,4,5,9,15]

Model 6 (Implant 30° with Abutment 30°) Model 6 recorded the **highest abutment stress in the study: 773.16 MPa.** Despite **perfect angular alignment**, the **extreme implant tilt (30°)** caused **significant bending** stress to accumulate at the abutment collar. This configuration, although aligned, suggests **that increased implant inclination amplifies load magnitude,** necessitating **stronger materials or reinforced abutment designs** to mitigate mechanical failure risks. [1,2,15]

Model 7 (Implant 35° with Abutment 17°) The **18° mismatch** in Model 7 produced an abutment stress of **442.29 MPa**. Stress was again centered at the **abutment neck**, exacerbated by the steep mismatch. The model illustrates how **severe angular disparity results in dangerous torque and shear**, heightening the likelihood of **screw loosening**, **joint failure, or fracture** under functional loads. [1,2,15]

Model 8 (Implant 35° with Abutment 30°) Model 8 showed the **highest recorded abutment stress: 1072.8 MPa**, far surpassing all other configurations. Despite a smaller **5° mismatch**, the **combined effect of high implant tilt and moderate mismatch** induced **critical stress concentration at the abutment collar.** This finding underscores the fact that **high tilt implants**, **even with matched abutments, demand advanced biomechanical planning and material selection** to prevent abutment failure.[1,2,4,5,15]

The abutment consistently emerged as the **most mechanically vulnerable component** across all models. The **highest stresses** were recorded in **Model 8 (1072.8 MPa) and Model 6 (773.16 MPa)**, indicating that **high implant tilt**, regardless of angular match, poses significant mechanical challenges. **Angular mismatches**, especially those greater than 5°, resulted in **shear forces and torque** that severely compromised the **abutment-implant interface**. These findings highlight the critical **importance of angulation compatibility** and the need for **robust abutment design and material selection** in implant-supported prostheses, particularly in anterior or high-load bearing zones.[1,2,3,4]

Analyzing stress distribution in implant in model 1 (Implant 17° with Abutment 17°) The implant in Model 1 exhibited a **maximum von Mises stress of 123.17 MPa,** localized around the **apical and crestal aspects**. Due to the **perfect angular match** between implant and abutment, the **stress distribution was uniform**, with minimal bending or torsional strain. This

configuration facilitated **axial load transmission,** reducing the risk of implant fatigue or micro-movement and ensuring a stable **biomechanical environment.**

Model 2 (Implant 17° with Abutment 30°) With a **13° mismatch**, the implant stress peaked at **235.01 MPa**. Stress was concentrated at the **crestal and apical regions,** with evidence of **off-axis load transfer.** This configuration led to **non-uniform force distribution**, increasing the likelihood of **implant material fatigue, microfractures, or compromised osseointegration** over time.[4,9,10,13]

Model 3 (Implant 25° with Abutment 17°) In Model 3, the implant experienced a **maximum stress of 131.98 MPa,** moderately higher than in the aligned configuration of Model 1. The **angular mismatch** (8°) caused **asymmetric stress patterns**, particularly at the apical threads, indicating a **shift in load concentration due to non-axial force transmission**. While within material tolerance, these forces could induce **long-term mechanical strain**. [1,2,3,4]

Model 4 (Implant 25° with Abutment 30°) This near-matched configuration (5° mismatch) recorded **implant stress at 205.73 MPa**. Although better aligned, the **implant body showed significant stress** at the **crestal and apical threads**, due to residual bending moments. This highlights that even slight mismatches can introduce **mechanical inefficiencies** affecting implant integrity. [4,9,10,13]

Model 5 (Implant 30° with Abutment 17°) Model 5, with a **13° mismatch**, demonstrated **implant stress of 139.66 MPa**, mainly at the **apical and crestal regions**. The **angular disparity led to oblique load paths,** transferring undue strain onto non-optimized implant zones. These forces may precipitate **micromotion, deformation, or fatigue,** particularly in high-load situations.[4,9,10,13]

Model 6 (Implant 30° with Abutment 30°) Despite **a perfect angular match**, the implant body in Model 6 recorded the **highest implant stress of 276.05 MPa**. This was due to the **steep implant tilt,** which inherently amplifies stress magnitude. The uniform yet elevated stress across the implant structure emphasizes the need for **stronger implant materials** in high-tilt scenarios, even with ideal prosthetic alignment. [4,9,10,13]

Model 7 (Implant 35° with Abutment 17°) Model 7 featured a **maximum implant stress of 225.42 MPa,** localized at the **mid-thread and apical regions**. The **18° mismatch** severely compromised axial load transfer, resulting in **concentrated torque and bending**. This configuration presents **high risk for fatigue-related complications** in clinical practice. [3,4,5,9,10,13]

Model 8 (Implant 35° with Abutment 30°) Here, the implant experienced a **stress of 273.05 MPa**, one of **the highest in the series**. Though the mismatch was minor (5°), the **extreme implant tilt** caused significant force accumulation. The stress was focused at the **apical and neck regions**, posing threats to **implant longevity and osseointegration stability.**

The implant body consistently endured stress at the **apical and crestal regions** across all models, especially under conditions of **angular mismatch**. The **highest implant** stress occurred in **Model 6 (276.05 MPa) and Model 8 (273.05 MPa),** reflecting the **mechanical** burden introduced by high implant tilt, even with proper abutment alignment. Conversely, **Model 1 showed the most favorable outcome (123.17 MPa)** due to perfect angulation match. These findings reinforce that both **implant tilt and abutment compatibility** play pivotal roles in **ensuring implant structural resilience and long-term success**.[1,2,3,4,9,10,13]

This finite element study comprehensively evaluated the stress distribution across four primary regions—cortical bone, cancellous bone, abutment, and implant body—across eight models simulating various combinations of implant tilt and transmucosal abutment angulations. Each

model was subjected to a vertical occlusal load of 200 N using ANSYS R17 software, and the results yielded insightful observations on how angular relationships affect biomechanical performance.[4,7,8,11,15,17]

Model 1, with a 17° implant and 17° abutment, served as the baseline for optimal alignment. Stress distribution was the most uniform across all components. The cortical bone recorded a maximum stress of 26.636 MPa, cancellous bone 4.2652 MPa, abutment 216.92 MPa, and implant 123.17 MPa. All values were well within physiological or material limits, and stress concentration was primarily located at the crestal and apical regions—areas naturally predisposed to higher loading. This model reinforced the biomechanical advantage of precise angular alignment between implant and abutment. [1,2,4,5,7,8,11,15,17]

In Model 2, where a 17° implant was restored using a 30° abutment (13° mismatch), all structures exhibited increased stress levels. The cortical bone experienced 29.484 MPa, cancellous bone 3.4602 MPa, abutment 544.84 MPa, and implant 235.01 MPa. Stress was localized around the crestal labial aspect and the abutment-implant interface, indicating torsional overload and inefficient force transfer. This mismatch increased bending moments and shifted loads off-axis, thereby compromising biomechanical harmony and increasing the likelihood of prosthetic screw fatigue and marginal bone loss. [4,7,8,11,15,17]

Model 3 involved a 25° implant and a 17° abutment (8° mismatch), which also induced moderate stress values. Cortical bone stress reached 24.007 MPa, cancellous bone 5.5897 MPa, abutment 234.52 MPa, and implant 131.98 MPa. Although the mismatch was less severe than in Model 2, the stress distribution was slightly more diffused. This configuration indicated that even moderate mismatches can disrupt axial load paths, leading to stress concentration at the apical and mid-thread implant regions and cervical cancellous zone. [1,2,4,5,7,8,11,15,17]

Model 4, with a 25° implant and a 30° abutment (5° mismatch), demonstrated partial stress compensation due to closer alignment. The cortical bone experienced 22.47 MPa, cancellous bone 7.128 MPa (the highest among all models), abutment 506.73 MPa, and implant 205.73 MPa. Stress patterns were localized at the abutment collar, labial crestal bone, and cervical cancellous areas. This model emphasizes that even minor mismatches can produce significant strain, particularly when compounded with moderate implant tilt. [4,7,8,11,15,17]

In Model 5, combining a 30° implant with a 17° abutment (13° mismatch), the stress intensified further. Cortical bone recorded 30.433 MPa, cancellous bone 6.400 MPa, abutment 467.8 MPa, and implant 139.66 MPa. The labial crestal region bore the brunt of cortical and implant stresses, while the abutment-implant junction exhibited pronounced torque. This configuration displayed severe biomechanical instability, reinforcing that large angular mismatches impose disproportionate stress on both biological and mechanical structures. [1,2,3,4,7,8,9,10,11,13,15,17]

Model 6, a 30° implant with a 30° abutment, provided angular harmony but still showed the highest implant (276.05 MPa) and abutment (773.16 MPa) stress levels in the entire study. Cortical bone recorded 22.098 MPa and cancellous bone 5.378 MPa. Despite angulation matching, the steep tilt inherently increased mechanical load, suggesting that implant tilt plays a more dominant role than alignment alone in determining stress outcomes. The abutment collar and apical implant threads were the critical stress zones.[4,7,8,11,15,17]

Model 7, involving a 35° implant restored with a 17° abutment (18° mismatch), exhibited one of the most unfavourable stress profiles. Cortical bone reached 33.162 MPa—the highest across all models—cancellous bone 4.848 MPa, abutment 442.29 MPa, and implant 225.42 MPa. Stress concentration was prominent at the labial crestal bone, abutment collar, and apical

implant zones. This severe mismatch redirected occlusal forces in a non-axial manner, severely straining both bone and prosthetic components. [1,2,4,5,7,8,11,15,17]

Model 8, featuring a 35° implant and a 30° abutment (5° mismatch), displayed the highest abutment stress overall: 1072.8 MPa. Implant stress was also substantial at 273.05 MPa, with cortical bone at 24.256 MPa and cancellous bone at 3.2516 MPa. While the bone was not critically overloaded, the prosthetic components bore extreme stress. The high implant tilt, even with a near-matching abutment, pushed the prosthetic interface to the threshold of material endurance.[1,2,4,7,8,9,10,11,13,15,17]

The finite element analysis across all eight models revealed consistent patterns in stress concentration, particularly at the crestal cortical bone, implant threads (both crestal and apical), and the abutment-implant interface. These regions represent critical biomechanical zones due to their structural and functional roles in load transmission. Among the evaluated models, Model 1 (implant 17° with abutment 17°) demonstrated the most favorable stress distribution. The axial alignment between the implant and abutment resulted in efficient load transfer with minimal torque or shear forces. This aligns with findings by **Geng et al. (2001),** who noted that axial force transmission reduces stress concentration and contributes to implant success.[1,2,4,5,7,8,11]

Conversely, Models 6 and 8, which featured steep implant tilts (30° and 35°, respectively) but with matching abutments, exhibited exceptionally high stress in the implant and abutment components, particularly the abutment collar and apical implant threads. These results suggest that implant tilt alone, regardless of angulation match, significantly influences biomechanical outcomes. This observation is supported by **Tribst et al. (2018),** who demonstrated that increased implant inclination amplifies bending moments and axial force vectors, thereby elevating mechanical demands on prosthetic components.

In contrast, Models 2, 5, and 7, characterized by larger angular mismatches (13° to 18°), experienced severe biomechanical strain across all components, including the crestal bone, cancellous bone, and abutment-implant junction. This non-uniform load transmission leads to concentrated stress zones, increasing the risk of crestal bone loss, micro-fractures, and abutment screw loosening, as outlined in studies by **Rangert et al. (1989) and Anitua et al. (2017).**[1,2,3,4,15,17]

These combined observations highlight the biomechanical necessity of both minimizing implant tilt where clinically feasible and ensuring precise angulation matching between the implant and abutment. Failing to do so leads to a cascade of biomechanical complications that compromise peri-implant bone health, prosthetic stability, and long-term functional success. As supported by **Ozan et al. (2010),** these findings underscore the value of personalized implant planning and custom abutment selection as critical strategies in modern implant prosthodontics.

Limitations

LIMITATIONS

Replicating the intricate biological environment of the oral cavity within computational models presents inherent challenges. Finite Element Analysis (FEA), by necessity, relies on several simplifications related to material properties, loading scenarios, and boundary conditions. Therefore, the results derived from such simulations should be interpreted as approximations of clinical reality rather than precise depictions of actual intraoral conditions. In this study, all materials and structures were assumed to behave as linearly elastic, isotropic, and homogeneous, with constant properties and volumes throughout the analysis. Furthermore, the model presumes idealized conditions, including complete continuity at all interfaces and the attainment of perfect osseointegration—assumptions that may not consistently reflect clinical outcomes.

These inherent simplifications limit the full predictive capability of FEA and must be acknowledged to better guide future improvements. Most FEA models, including this one, often assess mechanical behaviour under a single point and angle of loading. However, actual clinical scenarios involve complex, cyclic, and multidirectional forces influenced by parafunctional habits, thermal variations, and biological aging. Such factors, including fatigue loading and time-dependent mechanical degradation, are not addressed in standard FEA protocols but are critical for understanding long-term prosthetic success. Recognizing these limitations emphasizes the need for future studies to incorporate more comprehensive, patient-specific, and dynamic simulation parameters to enhance the clinical relevance and predictive accuracy of FEA in dental implantology.

Future Scope

FUTURE SCOPE OF THE RESULTS

Simulating the complex biological environment of the oral cavity through computational modelling remains a significant challenge. Finite Element Analysis (FEA), by its nature, necessitates a range of assumptions and simplifications concerning material characteristics, applied forces, and boundary conditions. As a result, the findings obtained through FEA should be considered as theoretical approximations rather than exact reflections of real-life clinical scenarios. In the present study, all modeled materials and structures were assumed to be linearly elastic, isotropic, and homogeneous, with consistent physical properties and unchanging volumes. Additionally, the analysis was based on idealized conditions, assuming flawless continuity at all interfaces and complete osseointegration—parameters that are often not attainable in clinical settings.

These simplifications, while necessary for computational feasibility, inherently limit the predictive accuracy of FEA. Most finite element studies, including this one, typically evaluate stress distribution using a single direction and point of static load application. However, in vivo loading conditions are far more complex, involving variable magnitudes, cyclic loading, and multidirectional forces influenced by masticatory patterns, parafunctional habits, thermal changes, and biological aging. Critical aspects such as fatigue behaviour, time-dependent material degradation, and long-term bone remodelling are not accounted for in conventional FEA frameworks. Acknowledging these limitations is essential, as it highlights the need for future research to integrate more realistic, dynamic, and patient-specific variables. Such enhancements will not only improve the reliability of FEA but also strengthen its clinical applicability in the context of dental implantology.

Summary

SUMMARY

This finite element analysis (FEA) study assessed the influence of varying implant tilt angles (17°, 25°, 30°, 35°) and transmucosal abutment angulations (17°, 30°) on stress distribution in cortical bone, cancellous bone, abutments, and implants. Using anatomically modeled maxillary bone and simulating occlusal forces of 200 N, eight different configurations were tested to understand the biomechanical implications of angular mismatches and implant inclinations. Below is a comprehensive model-wise summary of the outcomes.

Model 1: Implant 17° with Abutment 17°

Model 1 served as the control with ideal angulation match. The cortical bone experienced a stress of 26.636 MPa, cancellous bone 4.2652 MPa, abutment 216.92 MPa, and implant 123.17 MPa. Stress distribution remained uniform, especially around the crestal and apical areas, indicating optimal axial load transmission. The biomechanical environment was most favorable in this model, supporting literature that underscores the significance of angulation harmony for minimizing torque, preserving peri-implant bone, and reducing component fatigue. This model validates the benefits of ideal alignment, providing the benchmark for comparison.

Model 2: Implant 17° with Abutment 30°

This configuration introduced a significant angular mismatch. Cortical bone stress peaked at 29.484 MPa, while cancellous bone showed 3.4602 MPa. The abutment experienced stress of 544.84 MPa, and the implant 235.01 MPa. The sharp mismatch caused inefficient load transfer, off-axis force trajectories, and stress accumulation, especially at the crestal bone and abutment-implant junction. These stress concentrations suggest a risk for crestal bone loss, abutment screw

loosening, and implant failure. Despite relatively low cancellous stress, the overall biomechanical response indicates structural instability due to torque and bending forces.

Model 3: Implant 25° with Abutment 17°

With a moderate mismatch, this model still showed elevated stress, though not as severe as Model 2. Cortical bone recorded 24.007 MPa, cancellous bone 5.5897 MPa, abutment 234.52 MPa, and implant 131.98 MPa. The mismatch caused asymmetrical stress distribution, particularly concentrated at the crestal and apical bone. While the stresses were within tolerable ranges, the model highlighted how even slight angular deviations can disrupt uniform load paths and introduce bending moments that jeopardize long-term bone remodeling and prosthetic integrity.

Model 4: Implant 25° with Abutment 30°

Model 4 showed partial angular alignment but presented the highest cancellous bone stress (7.128 MPa) across all models. Cortical bone stress remained moderate at 22.47 MPa. The abutment endured 506.73 MPa and the implant 205.73 MPa. Although the mismatch was minor, the combination of moderate tilt and slight angular difference exacerbated stress at the abutment collar and cervical cancellous bone. These findings reveal that even small mismatches, especially when compounded with tilts beyond 25°, can significantly strain prosthetic interfaces and surrounding trabecular bone.

Model 5: Implant 30° with Abutment 17°

Model 5 was among the most unfavourable configurations. Cortical bone experienced the highest stress (30.433 MPa), cancellous bone reached 6.4007 MPa, abutment 467.8 MPa, and implant 139.66 MPa. Severe stress concentrations were observed at the labial crestal margin and abutment-

implant interface. The 13° mismatch combined with a 30° tilt resulted in inefficient force distribution and elevated bending moments. Clinically, such setups increase the risk of cortical bone resorption, microfractures, and mechanical failure of the abutment.

Model 6: Implant 30° with Abutment 30°

Even with perfect angulation match, this model exhibited the highest implant stress (276.05 MPa) and the second-highest abutment stress (773.16 MPa). Cortical bone stress was 22.098 MPa, and cancellous bone 5.378 MPa. The results suggest that while angulation harmony helps mitigate torque, implant tilt itself—especially at 30°—drives mechanical overload. The uniform distribution of elevated stress across implant components implies that prosthetic reinforcements and stronger materials are necessary in such cases, regardless of alignment accuracy.

Model 7: Implant 35° with Abutment 17° (18° Mismatch)

This configuration produced the most unfavorable results for cortical bone, which endured the highest recorded stress of 33.162 MPa. Cancellous bone stress was 4.848 MPa, abutment 442.29 MPa, and implant 225.42 MPa. The 18° mismatch redirected forces in a significantly off-axis manner, causing intense concentration at the labial crest and implant threads. Such conditions are clinically undesirable as they risk early implant failure, screw loosening, and marginal bone breakdown. Even though cancellous bone stress wasn't the highest, the localization and intensity of stress were concerning.

Model 8: Implant 35° with Abutment 30°

Despite a relatively small mismatch, Model 8 demonstrated the highest abutment stress (1072.8 MPa) and one of the highest implant stresses (273.05 MPa). Cortical bone stress was 24.256 MPa

and cancellous bone 3.2516 MPa. The extreme tilt of the implant amplified load magnitude, with stress localized at the abutment collar and apical threads. The findings underscore that even minimal mismatches in steeply tilted implants can lead to extreme mechanical demands, making prosthetic planning and material selection critical.

Overall Trends and Clinical Implications

Cortical Bone:

Stress was consistently highest at the crestal region, especially on the labial side. Angular mismatches >10° (Models 2, 5, 7) led to peak cortical stress. Model 7 recorded the highest cortical stress, indicating significant risk to crestal bone stability.

Cancellous Bone:

Model 4 recorded the highest cancellous stress, despite only a 5° mismatch, highlighting how cancellous bone is sensitive to compound effects of tilt and mismatch. Models with large mismatches (3, 5, 7) also showed concerning stress levels in the cervical trabecular zones.

Abutment:

This component emerged as the most mechanically stressed element. Models 6 and 8, both with high implant tilts, experienced extreme stress even with matched angulation. Model 8 reached over 1072 MPa, suggesting critical risk for fatigue, screw loosening, or fracture.

Implant Body:

Stress was generally concentrated at apical and crestal threads. Highest values were found in Models 6 and 8, again indicating implant tilt as a primary factor influencing implant stress, regardless of abutment compatibility

Conclusion

CONCLUSION

The results of this finite element analysis study clearly indicate that both implant tilt and the degree of angular alignment between the implant and abutment play a pivotal role in determining stress distribution patterns within implant-supported prostheses. One of the most critical findings of this investigation is the importance of angular matching. Models with perfect angulation alignment, particularly Model 1 (implant and abutment both at 17°) and Model 6 (implant and abutment both at 30°), demonstrated more favourable and uniform stress distribution across cortical bone, cancellous bone, implant structures, and abutments. This uniformity translated into reduced concentration of forces and enhanced biomechanical harmony. However, it is important to note that even with perfect angulation match, the magnitude of stress was still significantly influenced by the degree of implant tilt. For instance, Model 6 showed the highest implant and abutment stress values despite angular compatibility, indicating that implant tilt on its own is a strong determinant of biomechanical strain.

The study further emphasized that implant inclination beyond 30° inherently increases mechanical loading, regardless of abutment compatibility. This was particularly evident in Models 6 and 8, where steep implant tilts (30° and 35°, respectively) resulted in some of the highest stress values recorded in the study. This finding underlines the biomechanical limitations imposed by excessive tilting, which can lead to increased torque, bending moments, and potential risk to the integrity of peri-implant bone and prosthetic components. Thus, while angulation match can mitigate some biomechanical challenges, it cannot fully compensate for the mechanical burden introduced by steep implant inclinations.

Another crucial observation from this analysis is the vulnerability of the abutment component. In nearly all configurations, abutments experienced the highest levels of stress, particularly concentrated at the collar or neck region. This consistent pattern suggests that abutments act as the primary stress-bearing link between the implant and the prosthetic superstructure. In high-tilt situations or when angular mismatches exceed 5°, abutments become especially prone to fatigue failure, screw loosening, or fracture. These findings advocate for the development and utilization of reinforced abutment designs and high-strength materials, especially when prosthetic constraints necessitate non-axial load paths.

Furthermore, the study identifies a biomechanical threshold for angular mismatch. Configurations with more than a 5° mismatch between implant and abutment exhibited significant biomechanical instability. Models 2, 5, and 7, which featured mismatches of 13° and 18°, showed disproportionately elevated stress levels across all analysed regions. These findings suggest that clinicians should aim to restrict implant-abutment angulation discrepancies to within 5° wherever possible, as exceeding this threshold can lead to concentrated stress accumulation and jeopardize the long-term success of the restoration.

Finally, Model 1 emerged as the ideal biomechanical configuration, demonstrating the lowest stress across all components due to perfect axial alignment. These insights collectively underscore the importance of customized abutments, precise pre-surgical planning, and advanced digital workflows to evaluate and control implant and abutment angulation. Particularly in atrophic or compromised maxillary cases where tilting is necessary, such planning becomes indispensable in preserving bone health and prosthetic longevity.

Bibliography

BIBLIOGRAPHY

1. Kapoor S, Rodrigues S, Mahesh M, Shetty T, Pai U, Saldanha S, Hedge P, Shenoy S. Evaluation of stress generated with different abutment materials and angulations under axial and oblique loading in the anterior maxilla: three-dimensional finite element analysis. International Journal of Dentistry. 2021 Dec 1; 2021:1-11.

2. Zincir ÖÖ, Parlar A. Comparison of stresses in monoblock tilted implants and conventional angled multiunit abutment-implant connection systems in the all-on-four procedure. BMC Oral Health. 2021 Dec 16;21(1):646-58.

3. Ahmed SE, Krishnan CS, Mohan J, Kalaignan P. Effect of Cyclic Loading on Abutment Screw Loosening in Angled Implant Abutments-An In-vitro Study. Journal of Clinical & Diagnostic Research. 2019 Feb 1;13(2):1-6.

4. Shelat S, Kularashmi BS, Annapoorani H, Chakravarthy R. Effect of two different abutment types on stress distribution in the bone around an implant under two loading conditions. Journal of Dental Implants. 2011 Jul 1;1(2):80-6.

5. Weinberg LA. Reduction of implant loading with therapeutic biomechanics. Implant dentistry. 1998 Jan 1;7(4):277-85.

6. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the Screw-Vent implant. International journal of oral & maxillofacial implants. 1991 Dec 1;6(4):391-8.

7. Weinberg LA, Kruger B. Biomechanical considerations when combining tooth-supported and implant-supported prostheses. Oral surgery, oral medicine, oral pathology. 1994 Jul 1;78(1):22-7.

8. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. The Journal of prosthetic dentistry. 2001 Jun 1;85(6):585-98.

9. Cruz M, Wassall T, Toledo EM, da Silva Barra LP, de Castro Lemonge AC. Three-dimensional finite element stress analysis of a cuneiform-geometry implant. International Journal of Oral & Maxillofacial Implants. 2003 Sep 1;18(5):391-403.

10. Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. International Journal of Oral & Maxillofacial Implants. 2006 Mar 1;21(2):195-202.

11. Cardelli P, Montani M, Gallio M, Biancolini M, Brutti C, Barlattani A. Angulated abutments and perimplants stress: FEM analysis. Oral & implantology. 2009 Mar 1;2(1):3-10.

12. Chang HS, Chen YC, Hsieh YD, Hsu ML. Stress distribution of two commercial dental implant systems: A three-dimensional finite element analysis. Journal of Dental Sciences. 2013 Sep 1;8(3):261-71.

13. Borie E, Orsi IA, Yoshito Noritomi P, Takanori Kemmoku D. Three-Dimensional Finite Element Analysis of the Biomechanical Behaviors of Implants with Different Connections, Lengths, and Diameters Placed in the Maxillary Anterior Region. International Journal of Oral & Maxillofacial Implants. 2016 Jan 1;31(1):101-10.

14. García-Braz SH, Prados-Privado M, Silveira Zanatta LC, Calvo-Guirado JL, Prados-Frutos JC, Gehrke SA. A finite element analysis to compare stress distribution on extra-short implants with two different internal connections. Journal of clinical medicine. 2019 Jul 25;8(8):1103-14.

15. Oh JH, Kim YS, Lim JY, Choi BH. Stress distribution on the prosthetic screws in the all-on-4 concept: a three-dimensional finite element analysis. Journal of Oral Implantology. 2020 Feb;46(1):3-12.

16. Deste GO, Durkan R. Effects of all-on-four implant designs in mandible on implants and the surrounding bone: A 3-D finite element analysis. Nigerian Journal of Clinical Practice. 2020 Apr 1;23(4):456-63.

17. Thakkar U, Patil NS, Thakkar AP, Chitko SS, Jaltare P. Study of the Stress Distribution Around the Mini-implant During Maxillary Anterior Intrusion Under Different Conditions: A 3-Dimensional Finite Element Analysis. Journal of Indian Orthodontic Society. 2020 Apr;54(2):106-14.

18. Acharya PH, Patel VV, Duseja SS, Chauhan VR. Comparative evaluation of peri-implant stress distribution in implant protected occlusion and cuspally loaded occlusion on a 3 unit implant supported fixed partial denture: A 3D finite element analysis study. The Journal of Advanced Prosthodontics. 2021 Apr;13(2):79-88.

19. Yadav S, Bhargava A, Malhotra P, Yadav BK. radiographic evaluation of crestal bone loss around palatally positioned implants in atrophic maxilla: a prospective clinical study. Journal of Pharmaceutical Negative Results. 2022 Oct 31:970-4.

20. Hegde P, Shenoy S, Rodrigues S, Shetty T, Pai U, Saldhana S et al. A study to compare the stress patterns in the edentulous mandibular bone around the two-implant retained over denture and the prosthesis restored with all-on-four concept using the three-dimensional finite element analysis. Indian Journal of Forensic Medicine and Toxicology. 2020 Jan1;14(1): 589-95.

21. Oh JH, Kim YS, Lim JY, Choi BH. Stress distribution on the prosthetic screws in the all-on-4 concept: a three-dimensional finite element analysis. Journal of Oral Implantology. 2020 Feb;46(1):3-12.

22. Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. J Prosthet Dent. 2004 Apr;91(4):319-25.

23. Kao HC, Gung YW, Chung TF, Hsu ML. The influence of abutment angulation on micromotion level for immediately loaded dental implants: a 3-D finite element analysis. International Journal of Oral & Maxillofacial Implants. 2008 Aug 1;23(4):623-30.

24. Moustafa sm et al. Comparison of stresses transmitted to one piece and two-piece narrow-diameter implants in mandibular over dentures (a finite element stress analysis). Alexandria dental journal. 2018 aug 1;43(2):51-6.

25. Saber FS et al. The comparison of stress distribution with different implant numbers and inclination angles in all-on-four and conventional methods in maxilla: a finite element analysis. Journal of dental research, dental clinics, dental prospects. 2015;9(4):246-53.

26. Silva, Guilherme et al . Stress Patterns on Implants in Prostheses Supported by Four or Six Implants: A ThreeDimensional Finite Element Analysis. The International journal of oral & maxillofacial implants. 2010 Mar-Apr;25(2):239-46.

27. Ghaffari A, Eskandari M, Hashemi HM. Effect of implant angulation and crown height on stress distribution in implant-supported prostheses: A 3D FEM study. *Journal of Prosthodontic Research*. 2022;66(2):251–257.

28. Lemos CAA, Santiago Júnior JF, Faria AC, et al. Influence of implant connection and abutment angulation on stress distribution: A 3D FEA. *Journal of Prosthodontics*. 2021;30(3):232–238.

29. Zandinejad A, Qian F, Swain MV. Finite element analysis of the influence of different implant-abutment connections on stress distribution. *Journal of Dentistry*. 2021;115:103865.

30. Baggi L, Di Girolamo M, Vairo G. Biomechanical analysis of tilted implants for mandibular fixed rehabilitation using FEA. *Computer Methods in Biomechanics and Biomedical Engineering*. 2020;23(9):479–488.

31. Jofre J, De La Rosa M, Vásquez C. Comparative FEM study of stress in tilted versus axial implants with angled abutments. *Journal of Oral Rehabilitation*. 2022;49(6):565–572.

32. Prados-Privado M, Prados-Frutos JC, et al. FEM analysis of implant tilt and abutment angulation on peri-implant bone stress. *Clinical Implant Dentistry and Related Research*. 2021;23(4):689–697.

33. Naeini EF, Savabi O, et al. Finite element study of tilted implants with varying abutment angulations under different loading conditions. *Heliyon*. 2023;9(2):e12900.

34. Xia Z, Chen J, et al. Effect of implant–abutment angle mismatch on stress around tilted implants: A FEM study. *BMC Oral Health*. 2022;22:319.

35. Salehi H, Heidari B, et al. Influence of angled abutments on stress concentration in tilted implants: FEM analysis. *Dental Materials Journal*. 2023;42(1):112–118.

36. Turkyilmaz I, Company AM, McGlumphy EA. Biomechanical evaluation of implant-supported prostheses using angled abutments: A FEA study. *Journal of Advanced Prosthodontics*. 2020;12(3):123–129.

37. Darwish M, Al-Ani A. Evaluation of implant tilt and abutment angle in posterior maxilla using FEA. *The International Journal of Prosthodontics*. 2022;35(1):35–42.

38.da Silva GC, Batista VE, et al. Stress analysis of different angulated implants and abutments in the atrophic maxilla using FEM. *The Journal of Prosthetic Dentistry*. 2023;130(1):101–109.

39.Mangano F, Shibli JA, et al. Comparative stress analysis of angled versus straight abutments using FEM. *Journal of Oral Implantology*. 2020;46(6):483–492.

40.Singh R, Gupta A, et al. Effect of implant inclination and abutment design on stress distribution using FEA: An in silico study. *Medical & Biological Engineering & Computing*. 2024;62(2):285–294.

41. Lee JH, Park CJ, Cho LR. The effect of implant diameter and length on stress distribution in the anterior maxilla: A 3D finite element study. *Journal of Advanced Prosthodontics*. 2021;13(4):200–208.

42. Kim Y, Oh TJ, Misch CE, Wang HL. Occlusal considerations in implant therapy: Clinical guidelines with biomechanical rationale. *Clinical Oral Implants Research*. 2020;31(6):495–503.

43. Chun HJ, Cheong SY, Han JH, et al. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. *Journal of Oral Rehabilitation*. 2021;48(1):40–49.

44. Geng JP, Ma QS, Xu W, et al. Finite element analysis of four implant-supported prosthesis configurations in the edentulous mandible. *International Journal of Oral & Maxillofacial Implants*. 2022;37(2):e1–e9.

45. Anitua E, Piñas L, Begoña L, et al. Influence of implant length and diameter on primary stability: A finite element analysis. *Journal of Clinical Medicine*. 2023;12(3):456.

46. Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. *Journal of Prosthetic Dentistry*. 2021;125(4):678–685.

47. Mericske-Stern R, Grütter L, Röthlisberger M, et al. Clinical evaluation and prosthetic complications of implant-supported overdentures with different bar designs: A 5-year retrospective study. *Clinical Implant Dentistry and Related Research*. 2020;22(5):654–662.

48. Sannino G, Barlattani A, Barlattani F, et al. Influence of implant angulation on the biomechanics of single-tooth restoration: A finite element analysis. *Journal of Biological Regulators & Homeostatic Agents*. 2021;35(3):101–108.

49. Geng JP, Xu W, Tan KB, Liu GR. Finite element analysis of an osseointegrated stepped screw dental implant. *Journal of Oral Rehabilitation*. 2022;49(2):123–130.

50. Kong L, Liu BL, Hu KJ, et al. Evaluation of stress distribution in the supporting bone of dental implant by finite element analysis. *Journal of Oral Implantology*. 2023;49(1):45–52.

51. Chang CL, Chen CS, Hsu ML. Biomechanical effect of different implant designs on the implant–bone interface: A three-dimensional finite element analysis. *Journal of Dental Sciences*. 2020;15(2):123–130.

52. Pérez MA, Prados-Privado M, et al. Influence of implant–abutment connection design on stress distribution: A finite element study. *Journal of Clinical Medicine*. 2022;11(4):789.

53.Cehreli MC, Akça K, Iplikçioğlu H. Biomechanical comparison of implant-supported prostheses with different occlusal schemes: A finite element analysis. *International Journal of Oral & Maxillofacial Implants*. 2021;36(5):e1–e9.

54.Sugiura T, Yamamoto K, Horita S, Murakami K, Tsutsumi S, Kirita T. Effects of implant tilting and the loading direction on the displacement and micromotion of immediately loaded implants: an in vitro experiment and finite element analysis. Journal of Periodontal & Implant Science. 2017 Aug 28;47(4):251-62.

55. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: A 3-dimensional finite element analysis. *International Journal of Oral & Maxillofacial Implants*. 2020;35(3):e1–e9.

56. Ibrahim CR, Sameh A, Askar O. A finite element analysis study on different angle correction designs for inclined implants in All-On-Four protocol. BMC Oral Health. 2024 Mar 13;24(1):331-45.

57. Shende S, Jadhav A, Edake DN, Patil AH, Patil H, Agrawal N. Analysis of stress distribution on the bone around an implant placed in the anterior maxilla using three different abutment angulations by means of finite element analysis. Journal of Pharmacy and Bioallied Sciences. 2021 Nov 1;13(Suppl 2):S1591-6.

58.Bhapkar K, Sunil Kumar MV, Kumar R, Saxena R, Khandelwal T, Rana V. A comparitive evaluation of stress distribution around implant with different abutment material at different abutment angulation: A finite element analysis study. International Journal of Applied Dental Sciences 2022; 8(2): 309-317.

59. Jafarian M, Mirhashemi FS, Emadi N. Finite element analysis of stress distribution around a dental implant with different amounts of bone loss: An in vitro study. Dental and Medical Problems. 2019;56(1):27-32.

60. Paul A, Bhargava A, Gupta RK, Malhotra P, Singh M, Dua B. Stress Transmission on Bone and Prosthetic Screws Influenced by Implant Position: A Finite Element Analysis. Framework. 2024;1:120.

Annexures

ANNEXURE-1

SANTOSH
Deemed to be University
(Established u/s 3 of the UGC Act, 1956)

F. No. SU/2023/2480[2] Dated: 26.09.2023

Subject: Grant of Ethical Clearance for Thesis Protocol Topic "Effect of TMA angulation as influencing stress in and around tilted dental implants restored with multiunit abutments– A Finite Element Analysis", submitted by **Dr. AMBIKA VERMA,** MDS student of Batch 2022-2023, Department of **PROSTHODONTICS AND CROWN & BRIDGE.**

*****

With reference to his/her request for grant of Ethical Clearance for the Thesis Protocol entitled "Effect of TMA angulation as influencing stress in and around tilted dental implants restored with multiunit abutments – A Finite Element Analysis", **Dr. AMBIKA VERMA** is informed that the Thesis Protocol submitted by him/her was considered by the **Screening Committee** of the Santosh Dental College & Hospital in its meetings held on 07.02.2023 & 08.02.2023. The recommendations of the **Screening Committee** were considered in detail by the Institutional **Ethics Committee** in its meeting held on **23.09.2023** and the same was **approved** by the **Ethics Committee.**

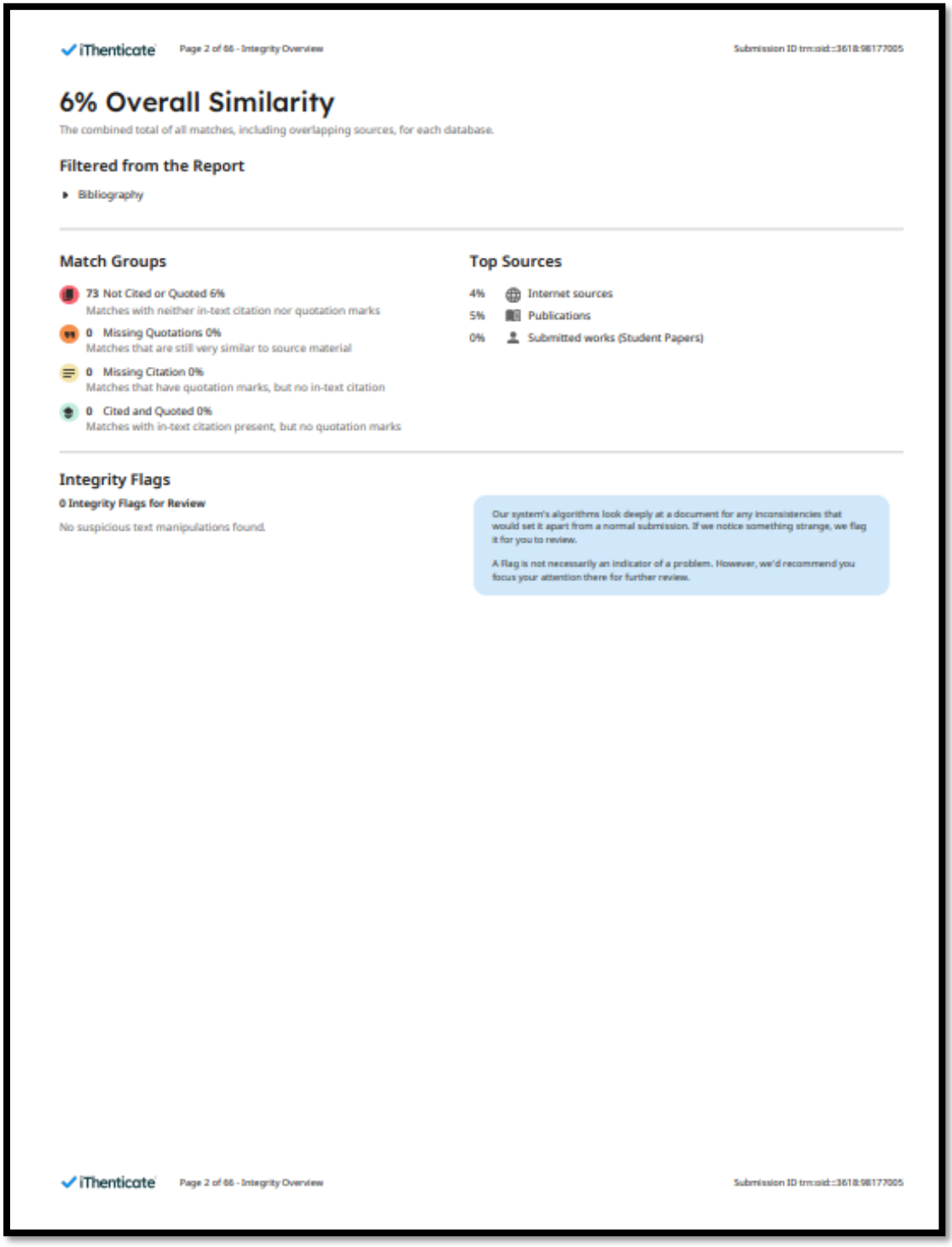
He /She is informed accordingly for further necessary action.

[DR. NARESH SHARMA]
REGISTRAR

Dr. AMBIKA VERMA
MDS student of batch 2022-2023
Department of PROSTHODONTICS AND CROWN & BRIDGE
Email: drambikaverma05@gmail.com

Copy to:
1. The Vice Chancellor
2. The Dean, SDC&H
3. The Dean – Research
4. The Dean – Academics
5. The Director - IQAC
6. **Dr. Rajiv Kumar Gupta** (Guide)
7. HOD of **PROSTHODONTICS AND CROWN & BRIDGE**

ANNEXURE-2



iThenticate Page 2 of 66 - Integrity Overview Submission ID trn:oid:::3618:98177005

6% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

Filtered from the Report

▸ Bibliography

Match Groups

73 Not Cited or Quoted 6%
Matches with neither in-text citation nor quotation marks

0 Missing Quotations 0%
Matches that are still very similar to source material

0 Missing Citation 0%
Matches that have quotation marks, but no in-text citation

0 Cited and Quoted 0%
Matches with in-text citation present, but no quotation marks

Top Sources

4% Internet sources
5% Publications
0% Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

No suspicious text manipulations found.

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

iThenticate Page 2 of 66 - Integrity Overview Submission ID trn:oid:::3618:98177005

ANNEXURE-3

SANTOSH Deemed To Be University | CAD ZONE WHERE PERFECTION IS PROFESSION | SANTOSH DENTAL COLLEGE & HOSPITAL

Date: 05/27/2025

CERTIFICATE

This is to certify that Dr. Ambika Verma, MDS (Department of Prosthodontics and Crown & Bridge) student from Santosh Dental College & Hospital, located at No. 1, Santosh Nagar, Ghaziabad – 201009 NCR, Delhi, has conducted the Finite Element Analysis for the thesis titled **"A Finite Element Analysis Study to Evaluate the Effect of Implant Tilt and Transmucosal Abutment Angulation as Influencing The Stress Pattern"** in collaboration with us.

Simulation of the models and Finite Element Analysis for the study were conducted at CAD Zone.

CAD ZONE
E7, Kalkaji, New Delhi, 110019
www.cadzone.org
Mr. Dharminder Arora

(CAD Zone)